

# Structural Optimization and Fatigue Life Assessment of Connecting Rod Using Topology Optimization and Finite Element Analysis

Mohamed Faisal<sup>1</sup> and Andrews Athisayam<sup>2\*</sup>

<sup>1</sup>m.mohamedfaisal2020@gmail.com, Department of Mechanical Engineering, National Engineering College, Kovilpatti, 628503, Tamil Nadu

<sup>2\*</sup>andrewsnzt@gmail.com, Department of Mechanical Engineering, National Engineering College, Kovilpatti, 628503, Tamil Nadu

**Abstract.** The connecting rod is a critical part of the engine to transfer forces between the piston and crankshaft by withstanding continuous cyclic loading. In this study, a standard connecting rod was modelled and analyzed using ANSYS for deformation, stress behavior, safety factor, and fatigue life. Duralumin was selected as the material, since it contains a high strength-to-weight ratio and good resistance to fatigue, making it suitable for lightweight reciprocating parts. Realistic boundary conditions were then applied by fixing the big end and applying an axial compressive load at the small end. Further topology optimization was performed to remove low-stress regions, which showed significant mass reduction. While higher stresses and a reduced safety factor were observed for the optimized model, the fatigue life remained the same due to the preservation of critical load paths. The final design has maintained an acceptable FoS, and using a slightly lower FoS is quite reasonable when weight reduction is of primary importance.

## 1 Introduction

A design optimization study by Aisha et al. [1] on diesel engine connecting rods illustrated how significant improvements in performance and durability can be achieved using appropriate design changes. Ali and Haneef [2] investigated the fatigue stresses acting on the connecting rod, particularly at the big end, showing that concentrated loads can critically influence failure behaviour. Chen et al. [3] considered topology optimization for fatigue resistance by penalizing cumulative fatigue damage; they were able to obtain superior structural life in continuum components. Gao et al. [4] presented a novel formulation for topological fatigue optimisation, taking into account material defect distributions to further enhance the reliability of the optimized structures. Jeong et al. [5] applied topology optimization for the investigation of fatigue and static failure behavior; their results showed that optimized designs reduce stress concentration. Another investigation by Jeong et al. [6] introduced variable-amplitude fatigue constraints using an equivalent static load approach in

---

\* Corresponding author: [andrewsnzt@gmail.com](mailto:andrewsnzt@gmail.com)

order to improve fatigue life predictions. Extending these latter investigations, Lee et al. [7] presented topology optimization within the frequency domain, as well as how dynamic loading conditions influence fatigue-driven design decisions. Rao et al. [8] performed fatigue analysis of connecting rods made from different materials, thus offering numerous pieces of information about material-dependent performance under various engine loads. Sayed and Lund [9] presented one of the early methodologies for structural optimization problems with constraints on fatigue life, laying the platform for most of the modern approaches for fatigue-based optimization. Sundar et al. [10] coupled design, analysis, and topology optimization for connecting rods to attain considerable weight reduction with no loss of structural rigidity. Zhang et al. [11] have carried out fatigue-based topology optimization under non-proportional loading, emphasizing the potential relevance of load path variability in structural performance. Finally, in work by Zhen et al. [12], a detailed failure analysis and manufacturing improvement study of cracking-type connecting rods were conducted, pointing out critical regions and making suggestions for process improvements to enhance reliability and sustainability.

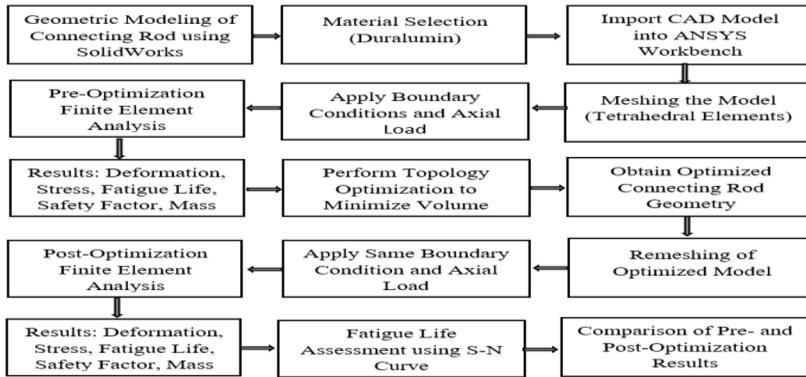
A connecting rod model was designed in accordance with the standard using SolidWorks, followed by analysis in ANSYS Workbench. The research study covers pre-optimization and post-optimization analyses: structural performance evaluation, fatigue life, and safety factor. This is mainly focused on achieving a drastic reduction in weight through topology optimization without affecting the structural reliability of the component. These results are presented through a comparison backed by graphical and tabular data in order to justify the optimization approach.

## 2 Methodology

The structural optimization and fatigue life assessment of the connecting rod were performed by adopting a structured simulation-based approach. This methodology was divided into a series of stages, including geometric modeling, choosing materials, meshing, applying boundary conditions, conducting pre-optimization analysis, topology optimization, post-optimization evaluation, and fatigue life assessment. The process began with geometric modeling in SolidWorks, with the formation of the connecting rod based on standard structural requirements. Then, the model was exported in IGES to make sure that it was compatible with ANSYS Workbench, which was used for optimizations.

The material to be used for the connecting rod was Duralumin, 2024-T3, aluminum alloy, and it has a high strength-to-weight ratio, very superior resistance to fatigue, and is light in weight. In ANSYS, the mechanical properties were given for Young's modulus, Poisson's ratio, yield strength, and ultimate tensile strength to precisely capture the material behavior under loading conditions. Material properties used in the analysis Table1.

The meshing was an important step in ensuring the accuracy of FEA. The model was discretized using tetrahedral elements, which effectively provide a good balance between the quality of the mesh and the feasibility of computation. In critical regions of stress concentration, the mesh is refined for better resolution of results with high accuracy. Also, the mesh independence was verified to check whether the refinement of mesh size did not affect the results much.



**Fig .1.** Process Flowchart of Structural Optimization and Fatigue Life Assessment of Connecting Rod

The meshing stage was crucial to ensuring the accuracy of finite element analysis (FEA). The model was discretized using tetrahedral elements, which provide an efficient balance between mesh quality and computational feasibility. A refined mesh was applied in critical stress concentration regions, ensuring better resolution and accuracy in the results. Mesh independence was checked to validate that the results remained consistent despite mesh refinements.

Before optimization, a static structural analysis was performed to evaluate the initial performance of the connecting rod. The model was subjected to boundary conditions, including fixed constraints on one end and axial loading on the opposite end to simulate operating conditions. The pre-optimization FEA results provided critical data on deformation, stress distribution, fatigue life, safety factor, and mass, forming the baseline for further optimization.

After assessing the initial performance, topology optimization was performed to reduce the mass of the connecting rod while maintaining structural integrity. The optimization process in ANSYS Workbench follows a systematic principle where regions with minimal contribution to load-bearing capacity are removed. The objective function was defined as mass minimization, with stress constraints ensuring the design retains adequate mechanical strength. The optimization algorithm iteratively removes inefficient material based on stress distribution and load paths, leading to a structurally optimized geometry.

After obtaining the optimized design, the new geometry was modeled and remeshed using tetrahedral elements to maintain simulation accuracy. Similarly, the post-optimization model was prepared to conduct the FEA under the same boundary conditions and axial loads as in the pre-optimization stage. The refined model was further analyzed for its deformation, stress distribution, fatigue life, and safety factor issues to confirm the performance of the optimized model. Fatigue analysis was conducted to examine the durability of the optimized connecting rod under repeated loading. The S-N curve method had been used to relate the stress amplitude with the number of cycles to failure. The result confirms that the optimized design will be able to endure cyclic stresses without early failure during its service life. A comparison was conducted to judge the improvements achieved through topology optimization. Improvement in key performance parameters is noted in terms of mass reduction, deformation, stress distribution, safety factor, and fatigue life.

**Table 1.** The material properties of Duralumin

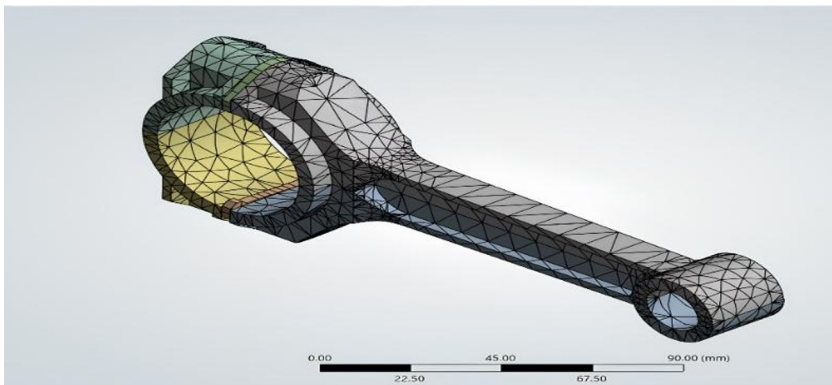
Property	Value
Density ( $\text{kg/m}^3$ )	2770
Young's Modulus (MPa)	72400
Poisson's Ratio	0.33
Yield Strength (MPa)	324

### 3 Meshing Strategy

The geometric model of the connecting rod was imported into ANSYS Workbench, where a high-quality mesh was generated to capture the structural behaviour under various loading conditions is shown in Fig 2. The meshing was performed using Tetrahedral elements, which are highly suitable for complex geometries like the connecting rod due to their flexibility in fitting irregular shapes.

The element size was carefully selected to optimally balance both computational efficiency and the accuracy of the results. The global element size used was approximately 5 mm with further mesh refinement applied in those regions where stress concentration was highly probable: that is, the small end, big end, and transition areas. Refinement would ensure the mesh was sufficiently dense to resolve localized variations in stresses without undue increases in computational cost.

The mesh in the pre-optimization model contained approximately 6486 elements and 13516 nodes.

**Fig 2.** Meshing of Connecting Rod Before Topology Optimization

### 4 Boundary Conditions

Boundary conditions have been applied in an effort to closely replicate the mechanical behaviour of the connecting rod during an engine cycle. Real operation constraints and component load paths have been considered; therefore, a fixed support was assigned to the big end, as discussed in Fig. 3(A). These conditions will not allow translation of the big end along all three axes of its current position. In an engine, that region would remain inside the

crankshaft's journal bearing, which restricts linear motion and permits rotation. This simulation is not able to model the bearing rotation; however, for the purposes of the peak loading scenario being simulated, the fixed support provides a close approximation to the stabilizing constraint provided by the crankshaft.

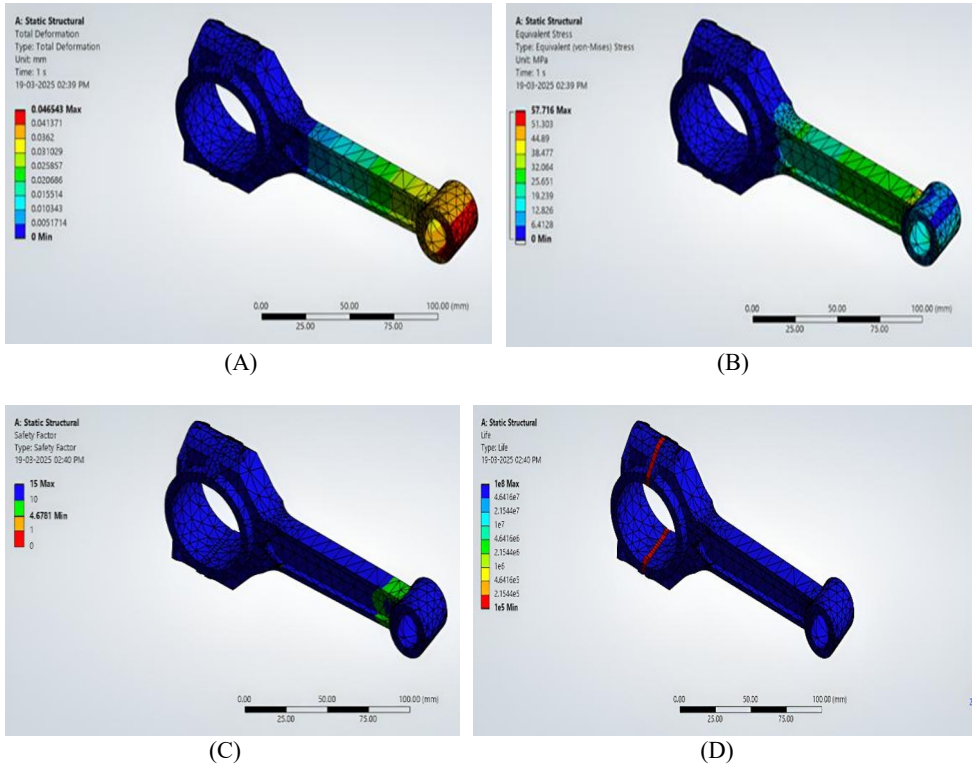
In Fig. 3(B), an axial compressive force of  $-3933.9$  N was placed along the longitudinal axis of the rod at the small end. This load is indicative of the downward force transferred from the piston during the combustion stroke. This direction and magnitude of this applied load model the highest compressive phase experienced by the connecting rod. This model will, therefore, have the capability to realistically simulate the stress distribution, deformation behaviour, and fatigue response that this component undergoes during engine operation by adopting a model which combines a fixed constraint at the big end with a realistic axial load at the small end. These conditions ensure that the outputs of analyses most closely represent the mechanical demands encountered in service.



**Fig 3.** Boundary Conditions, (A) fixed support applied at the big end, (B) axial force applied at the small end

## 5 Pre-Optimization Analysis

Pre-optimization structural analysis has been carried out to assess the baseline performance of the original connecting rod model. Total deformation, equivalent von Mises stress, safety factor, and fatigue life under cyclic loading are the key performance indicators that have been studied. The total deformation under the static loading condition turns out to be pretty small, with a peak value of  $0.0465$  mm only, as depicted in Fig 4(A). It can easily be realized from the von Mises distribution that the maximum stress happens in the transition zone between the shank and the big end of the connecting rod. The peak value is  $57.716$  MPa, which is far below the yield strength of the material  $324$  MPa. As shown in Fig 4(B), Fig. 4(C) shows the distribution of the safety factor in the model. The minimum value of the safety factor has been evaluated to be  $4.6781$ . Fatigue life distribution, as shown in Fig 4(D) indicates that those lowest life regions that correspond to red color belong to the stress concentration zones at the big end.



**Fig 4.** Pre-Optimization Analysis, (A) Total Deformation, (B) Equivalent von Mises Stress, (C) Safety Factor, (D) Fatigue Life

Results are summarized in Table 2. Pre-optimization results showed that stress and deformation are comparatively low, while the high factor of safety ensures the reliability of the structure. The fatigue life of the connecting rod is about 9.3 million cycles, confirming that it is able to bear cyclic loading over extended periods.

**Table 2.** Pre-Optimization Analysis Results

Parameter	Value
Total Deformation (mm)	0.0465
Equivalent von Mises Stress (MPa)	57.716
Safety Factor	4.6781
Fatigue Life (Cycles)	$\sim 9.3 \times 10^6$ cycles
Mass (grams)	223.37

## 6 Topology Optimization

Topology optimization using ANSYS Workbench has been performed to reduce the connecting rod's weight without compromising any of its structural integrity. The objective for this optimization has been set to minimize the volume while maintaining the safety factor greater than the acceptable limit. This optimized model targeted material removal from low-stress regions, especially in the shank area of the connecting rod presented in Fig 5.

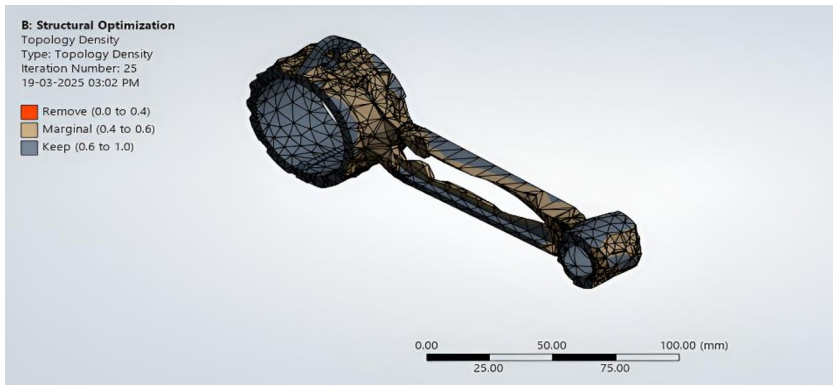


Fig 5. Topology Optimization Result

This optimization was able to reduce the volume by as much as 26.57% after multiple iterations. The mass of the connecting rod went down from 223.37 grams to 164.06 grams, showing a great improvement in the strength-to-weight ratio.

## 7 Post Optimization Analysis

After topology optimization was performed and the low-stress material regions removed, this optimized model was remeshed to approximately 6535 elements with 13146 nodes. The quality of the mesh remains good, as does the resolution, to maintain accuracy in post-optimization analysis displayed in Fig 6.

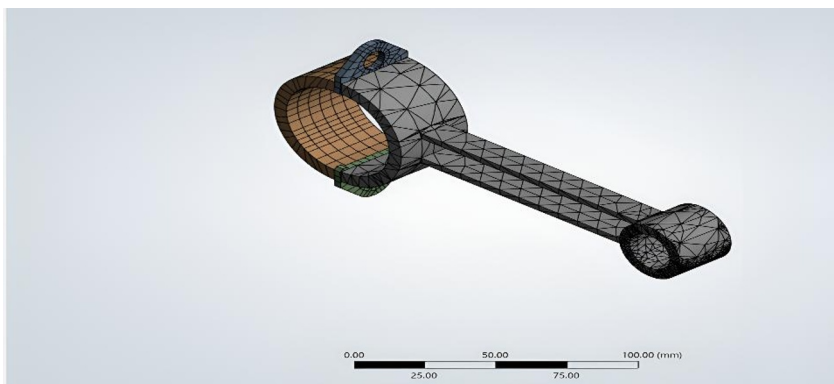
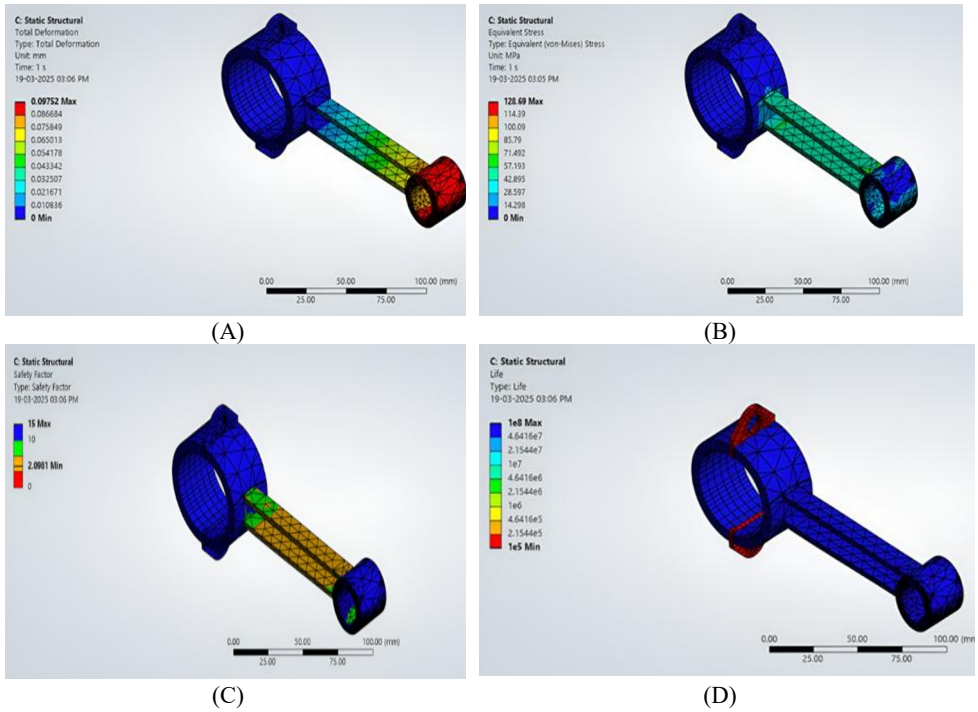


Fig 6. Meshing of Connecting Rod After Topology Optimization

The post-optimization deformation increased to 0.0975 mm, over twice the pre-optimized value. The change is graphically shown in Fig 7(A), with peak displacement at the loaded end. The maximum equivalent von Mises stress became 128.69 MPa, as seen in Fig 7(B). The safety factor contour plot Fig 7(C) showed a minimum of 2.0981. The fatigue life distribution in Fig 7(D) indicated that the minimum fatigue life remained well within the least of the pre-optimized model.

The modified model of the connecting rod underwent the same structural analysis as the original to assess its performance. Results obtained after optimization are summarized in Table 3.



**Fig. 7.** Post Optimization Analysis, (A) Total Deformation, (B) Equivalent von Mises Stress, (C) Safety Factor, (D) Fatigue Life

Though increased deformation and stress values were found, the optimized design maintains a safety factor greater than critical limits and the fatigue life remains constant. An effective reduction in mass could be achieved by the optimized design to improve efficiency without any loss in structural performance.

**Table 3.** Post-Optimization Analysis Results

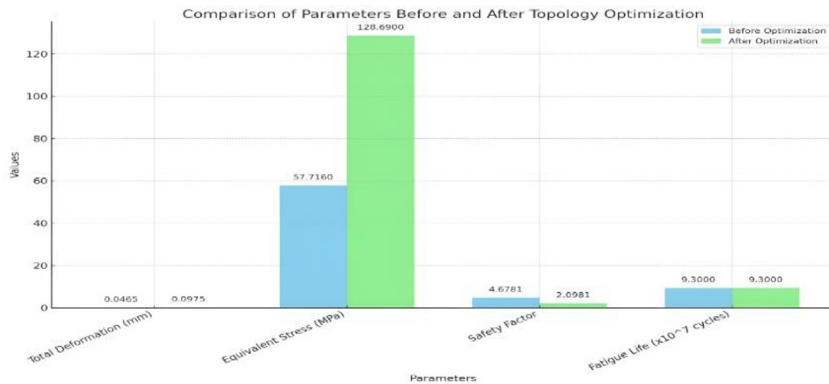
Parameter	Value
Total Deformation (mm)	0.0975
Equivalent von Mises Stress (MPa)	128.69
Safety Factor	2.0981
Fatigue Life (Cycles)	$\sim 9.3 \times 10^6$ cycles
Mass (grams)	164.06

## 8 Results

Further detail of how removing low-stressed material has affected the structural response of the component as a whole is given by the comparative evaluation of the connecting rod before and after topology optimization. From its deformation, stress distribution, and safety factor to the change in mass, all these differences of the optimized model are represented in Table 4 from the original design. These changes would suggest how the load path has concentrated at the removal of excessive material. Such an analysis would also reveal how redistribution of stresses leads to a change in mechanical behavior for the connecting rod under identical loading conditions.

**Table 4.** Comparative Analysis of Pre and Post- Optimization Results

Parameter	Before Optimization	After Optimization	Percentage Change
Total Deformation	0.0465	0.0975	+109.68
Equivalent von Mises Stress (MPa)	57.716	128.69	+122.91
Safety Factor	4.6781	2.0981	-55.15
Fatigue Life (Cycles)	$9.3 \times 10^4$	$9.3 \times 10^4$	0
Mass (Grams)	223.37	164.06	-26.57



**Fig 8.** Comparison Bar Graph of Pre- and Post-Optimization Results

From the bar graph, it can easily be seen that the topology optimization results in improved weight reduction. Even though both the deformation and stress levels increased, the optimized connecting rod still satisfies all requirements on safety. The safety factor decreases by about 55%, but is still above the critical threshold to provide for reliable operation at the applied loading conditions.

The most important result of this optimization is the 26.57% reduction in mass. This is favorable for automotive applications, as reducing the mass of such reciprocating parts as connecting rods contributes to improved fuel economy, a reduction of the forces of inertia, and generally better engine performance.

Further confirmation of reliability for the optimized design is seen from the fatigue life analysis. Although the safety factor had reduced, the levels of fatigue life maintained steady

at approximately 9.3 million cycles. This verifies that the optimized model retains its durability under cyclic loading conditions and hence justifies the effectiveness of the optimization process.

## 9 Conclusion

The work reported herein presents the use of topology optimization in pursuit of improved structural efficiency in an internal combustion engine connecting rod. FEA used in conjunction with the optimization techniques allowed component mass reduction by 26.57%, reducing it from 223.37 g to 164.06 g without sacrificing major mechanical performance characteristics. However, there was increased deformation and stress in the optimized model, while the factor of safety, as calculated for the connecting rod, fell within acceptable limits at 2.0981 under normal automotive operating conditions.

Another observation that might be made from this analysis is that, after mass reduction, the fatigue life remained at about 9.3 million cycles. Most of the material was taken away from the low-stressed part of the structure during optimization; it did not affect the critical zones of load-carrying and, consequently, the amplitude of the working stress responsible for the fatigue failure remained almost unchanged. Of course, the reduction in factor of safety can be understood, as less material is available to share the same external load, which may result in an increase in local stresses. Nevertheless, a slightly smaller factor of safety can be acceptable when the design is able to meet minimum requirements in terms of reliability and performance-focused or efficiency-driven applications.

This optimized connecting rod will have the advantage of a reduction in inertia forces and an improvement in the efficiency of the engine; hence, it is suitable for practical application. Further work could be devoted to experimental testing, further refinement of the constraints on optimization, and studies of advanced materials for realizing further improvements in performance.

## References

1. Aisha, M., Mohammed A.H. Ali, M., Shanono. S., Design optimization of a diesel connecting rod, materials today, Proceedings Volume **22**, Part 4, pp. 1600-1609, 2020.
2. Ali, H., Hane, M., Analysis of Fatigue Stresses on Connecting Rod Subjected to Concentrated Loads At the Big End, materials today Proceedings., Volume **2**, pp. 2094-2103, Issues 4-5, 2015.
3. Chen, S., Kai Long, Y., Pin Wen, S., Nouman, M., Fatigue-resistance topology optimization of continuum structure by penalizing the cumulative fatigue damage, Advances in Engineering Software Volume **150**, pp. 1652-1685, December 2020.
4. Gao, J., Caivano, M., Tridello, C., Chiandussi, C., Haitao Ma, V., Paolino, F., Berto, S., Innovative formulation for topological fatigue optimisation based on material defects distribution and Topfat algorithm, International Journal of Fatigue Volume **147**, pp. 1547-1604 June 2021.

5. Jeong, J., Choi, S., Yoon., Fatigue and static failure considerations using a topology optimization method, *Applied Mathematical Modelling* Volume **39**, pp. 1137-1162, Issues 3–4, February 2015.
6. Jeong, J., Lee, C., Choi, S., Topology optimization considering the fatigue constraint of variable amplitude load based on the equivalent static load approach, *Applied Mathematical Modelling*, Volume **56**, pp.626-647, April 2018.
7. Lee, C., Yoon, B., Jeong, G., Topology optimization considering fatigue life in the frequency domain, *Computers & Mathematics with Applications*, Volume **70**, pp. 1852-1877, Issues 8, October 2015.
8. Rao, S.V., T.V. Hanumanta Rao, T.V., Satyanarayana, K., Nagaraju, B., Fatigue analysis of sundry I.C engine connecting rods, *materials today Proceedings* Volume **5**, pp. 4958-4964, , Issue 2, Part 1, 2018.
9. Sayed, M., Lund, E., Structural optimization with fatigue life constraints, *Engineering Fracture Mechanics*, Volume **37**, pp. 1149-1156, , Issue 6, 1990.
10. Sundar, G., dharan, M., Vishwa, D., Kumar. A., Design, analysis and topology optimization of connecting rod, *materials today Proceedings* Volume **46**, Part 9, pp. 3430-3438, 2021.
11. Zhang, S., Chau Le Z., Gain, L., Julián A., Fatigue-based topology optimization with non-proportional loads, *Computer Methods in Applied Mechanics and Engineering* Volume **345**, , pp. 805-825, 1 March 2019.
12. Zhen, S., Jiao G., Xiwei Lv, C., Ding, S., Study of failure analysis and manufacturing processes improvement of cracking connecting rod, *Journal of Mechanical Science and Technology* Volume **57** pp. 5445-5453, 11, November 2022