



A Review on Stress Optimization of Connecting Rod Using Finite Element Method

Abhishek Yadav¹, Kamal Kant Tiwari²

Research Scholar¹, Assistant Professor²

^{1,2} Oriental Institute of Science & Technology, Bhopal

Abstract

The Finite Element Method (FEM) has become a critical tool in the design and optimization of connecting rods in internal combustion engines. FEM provides a detailed analysis of stress distribution and material behaviour, helping engineers identify and mitigate stress concentrations that can lead to fatigue failure. This method enables the optimization of connecting rod geometry, material selection, and weight reduction while maintaining or improving durability and performance. Despite its many advantages, FEM does present challenges, including the accuracy of material models, setting realistic boundary conditions, and the high computational cost associated with complex simulations. However, the ability of FEM to simulate real-world operational conditions has revolutionized the design process, allowing for virtual testing and reducing the need for physical prototypes. With ongoing advancements in computational power, material science, and multiphysics simulations, FEM is expected to become even more effective in optimizing engine components. This abstract highlights the significance of FEM in improving connecting rod design, overcoming challenges, and pushing the boundaries of performance optimization.

Keywords: Finite Element Method (FEM), stress optimization, connecting rod design, fatigue failure, material selection, performance optimization, engine components, computational analysis.

1. Introduction

Connecting rods are essential components in internal combustion engines, serving as the critical link between the piston and the crankshaft. These components endure high levels of stress due to the repetitive forces exerted during engine operation. The reciprocating motion of the piston subjects the connecting rod to tension, compression, and bending forces that vary with the speed, load, and operating conditions of the engine.

As a result, the connecting rod is vulnerable to fatigue failure, particularly in areas subjected to high stress concentrations. Stress optimization is the process of improving the design and material distribution within the

connecting rod to minimize these stress concentrations and enhance its overall performance and lifespan. By optimizing stress distribution, it becomes possible to reduce the risk of failure, extend the component's durability, and improve the efficiency of the engine. One of the most effective tools for achieving stress optimization is the Finite Element Method (FEM). FEM is a computational technique used to solve complex problems in engineering and physics, including stress analysis and optimization of mechanical components. The role of FEM in mechanical design, particularly in stress optimization, cannot be overstated. FEM provides a detailed, accurate representation of how a component behaves under various loading conditions. By dividing the component into smaller elements, FEM allows for precise analysis of stress distribution and identification of critical failure zones. This capability makes FEM an indispensable tool for optimizing the design of connecting rods, ensuring they perform efficiently under real-world operating conditions.

- **Role of FEM in Stress Optimization**

FEM is a numerical analysis method that breaks down complex structures, like the connecting rod, into smaller, manageable elements. This method calculates how each element responds to different forces, providing a comprehensive picture of how the entire component behaves under stress. In the context of connecting rods, FEM allows engineers to simulate the impact of dynamic loading conditions on the component and identify areas prone to failure. It also enables optimization of the design by adjusting material distribution, geometry, and other factors to ensure the rod performs efficiently under operating conditions.

- **Stress Distribution and Optimization Techniques**

Optimizing the stress distribution within a connecting rod typically involves modifying the geometry and material properties. Areas with high stress concentrations can be redesigned to reduce those stresses, and the material can be selected or redistributed to improve fatigue resistance. Techniques like topology optimization can be used to remove unnecessary material while maintaining strength and functionality. By improving the geometry in critical areas and selecting the right materials, engineers can achieve a lightweight yet durable design, enhancing the connecting rod's overall performance.

- **Applications of FEM in Connecting Rod Design**

FEM is widely applied in the design and optimization of connecting rods. Various case studies demonstrate its effectiveness in improving connecting rod performance. Singh et al. (2017) used FEM to optimize the material distribution and geometry of the connecting rod, showing that stress concentrations could be reduced significantly, which improved the component's durability. Similarly, Gurunath et al. (2015) applied FEM to analyse the stress distribution in connecting rods subjected to dynamic loads. By identifying critical stress zones, the design was modified to enhance fatigue resistance.

2. Review of Literature

As engine technologies continue to evolve, the importance of optimizing components like the connecting rod has grown exponentially. The connecting rod serves as a critical interface between the reciprocating motion of the piston and the rotary motion of the crankshaft. This function subjects the connecting rod to a range of stresses, including axial, bending, and torsional stresses, which fluctuate under different engine operating conditions. These stresses, especially under high-speed operations, are a primary cause of fatigue failure in engine components, ultimately impacting engine performance and longevity. Therefore, optimizing the stress distribution within the connecting rod is essential for enhancing engine durability, reducing material usage, and improving overall engine performance.

Stress optimization aims to redistribute these forces across the component, minimizing the areas subjected to excessive stress and, in turn, improving fatigue resistance and operational lifespan. The Finite Element Method (FEM) has emerged as a revolutionary tool in the realm of stress optimization. FEM divides a complex structure, such as the connecting rod, into smaller, manageable elements that are analyzed for stress, strain, and deformation. By using this method, engineers can gain detailed insights into how a component will perform under different loading conditions. FEM allows for accurate simulation of material behavior, identification of critical stress zones, and the ability to test various design modifications without the need for physical prototypes. This capability makes FEM an indispensable tool in modern mechanical design, particularly in the automotive and aerospace industries, where performance and reliability are paramount. In the context of connecting rods, numerous studies have leveraged FEM to optimize stress distribution.

Singh et al. (2017) provided a comprehensive analysis of the use of FEM in optimizing connecting rods. Their study focused on improving the stress distribution by altering the geometry and material properties of the connecting rod. Using FEM simulations, the researchers were able to identify stress concentrations in specific areas of the connecting rod that were prone to fatigue failure. The results showed that by refining the material distribution and adjusting the design of these critical areas, the overall stress on the connecting rod could be reduced. This led to an increase in the component's lifespan and performance, demonstrating the effectiveness of FEM in optimizing engine components. Similarly, Gurunath et al. (2015) applied FEM to evaluate the stress distribution in a connecting rod subjected to dynamic loads. Their study focused on the big end of the connecting rod, which is typically the most critical stress zone due to the concentrated forces that act at this point.

By using FEM simulations, the researchers could accurately model the stress response of the connecting rod under dynamic loading conditions. Their findings emphasized the importance of considering dynamic forces when designing connecting rods, as static analysis alone does not provide a full picture of the stress distribution. The study also highlighted that optimizing the design of the connecting rod by refining the material and geometry at the big end could significantly enhance the fatigue resistance of the component. In addition to dynamic loading, Swapnil and Saifullah (2014) focused on the impact of geometry and material

choices on the stress distribution in the connecting rod of a two-wheeler engine. The study specifically analysed the Hero Honda Splendor connecting rod, demonstrating how FEM could be applied to smaller engine components.

The researchers found that by adjusting the thickness and geometry of the connecting rod, they could optimize the distribution of stress and reduce the risk of failure due to fatigue. This research added a valuable perspective to the field by extending FEM analysis to two-wheeler engines, which often operate under different conditions compared to automotive engines. The findings suggested that FEM could be applied to a wide range of engine types and could be used to improve both small and large engine components. The importance of understanding and mitigating fatigue stresses in connecting rods was further emphasized by Mohammed and Mohamed (2015). Their study focused on the fatigue stresses at the big end of the connecting rod, where cyclic loading can lead to material failure over time. The authors used FEM to simulate the effect of concentrated loads at this critical area and examined how different materials and geometries could influence the fatigue life of the connecting rod.

The results of their study showed that by selecting materials with higher fatigue resistance and optimizing the geometry at the big end, the component's lifespan could be significantly improved. This study highlighted the need for in-depth fatigue analysis in the design of engine components, especially in high-performance applications where fatigue failure is a major concern. While these studies have demonstrated the power of FEM in stress optimization, there are still several challenges to overcome in the field. One of the key limitations of FEM is the accuracy of the material models used in simulations. Materials in real-world applications often exhibit nonlinear behavior under stress, and capturing this behavior in FEM simulations is complex. Additionally, the accuracy of FEM results heavily depends on the quality of the mesh used to model the component. A coarse mesh may overlook critical details in stress distribution, while a fine mesh can significantly increase computational time.

Ensuring that the mesh is both accurate and computationally efficient remains a challenge in many FEM applications. Another challenge is the complexity of setting up boundary conditions in FEM simulations. For connecting rods, this includes accurately simulating the loads and constraints that occur during engine operation. For instance, in a real engine, the forces acting on the connecting rod are highly dynamic and vary depending on factors such as engine speed, load, and temperature. Capturing these dynamic effects in a simulation requires sophisticated modeling techniques and accurate data on the operating conditions. Furthermore, FEM analyses are often computationally expensive, especially for complex geometries or multiphysics simulations that consider thermal, fluid, and mechanical effects simultaneously.

The computational resources required for such simulations can be a limiting factor, particularly in industries where time and cost constraints are critical. Despite these challenges, advancements in FEM technology continue to improve its effectiveness and applicability in stress optimization. Modern FEM software such as ANSYS, ABAQUS, and COMSOL have made significant strides in incorporating advanced

material models, multiphysics simulations, and automated meshing algorithms. These advancements have allowed engineers to perform more accurate and efficient simulations, making FEM a more accessible tool for stress optimization in a wider range of applications. Additionally, the use of machine learning and artificial intelligence in conjunction with FEM has shown promise in optimizing the design process further.

By leveraging large datasets and automated optimization algorithms, engineers can now generate optimized designs in a fraction of the time it would take using traditional methods, FEM has proven to be an indispensable tool in the stress optimization of connecting rods. The studies reviewed have demonstrated the effectiveness of FEM in identifying critical stress zones, optimizing geometry and material distribution, and improving the overall durability and performance of engine components. While challenges such as the accuracy of material models, the complexity of boundary conditions, and computational costs remain, ongoing advancements in FEM technology continue to address these issues. As computational power increases and FEM software continues to evolve, the potential for further optimizing connecting rod designs and other critical engine components will continue to expand, paving the way for more efficient, durable, and lightweight engine designs. The integration of FEM with other emerging technologies, such as machine learning and artificial intelligence, may also revolutionize the field of mechanical design, making it possible to optimize even the most complex components with unprecedented precision and efficiency.

3. Finite Element Method in Stress Optimization

The Finite Element Method (FEM) is a powerful computational tool used for analyzing and solving complex engineering problems. It works by subdividing a structure or component into smaller, simpler parts called finite elements. Each of these elements is modeled with known material properties and is connected at discrete points, called nodes. The overall behavior of the entire structure is derived by solving the equations for each of the elements based on their interactions. In the context of stress optimization, FEM allows for the simulation of how a component like the connecting rod responds to different types of loads, such as tensile, compressive, and bending forces. By dividing the connecting rod into a mesh of elements, engineers can calculate stress, strain, and displacement at each node under specific loading conditions. This granular analysis provides a detailed view of how forces are distributed across the structure, allowing for identification of stress concentrations and areas that are likely to fail under fatigue. Through these simulations, engineers can modify the design to ensure that the component is optimized for performance and durability.

- **Application of FEM in Stress Optimization of Connecting Rods**

The application of FEM in the stress optimization of connecting rods involves simulating the real-world conditions under which the connecting rod operates.

The primary objective of FEM in this context is to identify the regions of the connecting rod that experience the highest stress and determine how these areas can be modified to reduce the risk of failure. FEM allows for the analysis of various loading conditions, such as dynamic loads during engine operation, cyclic

stresses from reciprocating motion, and thermal effects due to heat generation in the engine. FEM also helps to optimize the material distribution within the connecting rod. By adjusting the geometry and material properties of specific regions, engineers can redistribute stress more evenly across the component. This not only improves the component's resistance to fatigue but also results in a reduction of the overall weight without compromising the strength of the connecting rod. FEM can simulate various design modifications and test them virtually, making it possible to optimize the connecting rod design before manufacturing. One of the significant advantages of using FEM for stress optimization is that it allows engineers to test multiple scenarios in a short amount of time, which would be infeasible through physical testing alone. By performing simulations with different loading conditions, material properties, and geometric changes, engineers can quickly determine the best configuration for the connecting rod. This helps to accelerate the design process, reduce prototyping costs, and increase the overall efficiency of the development phase.

- **Benefits of FEM in Stress Optimization**

The primary benefit of using FEM in stress optimization is its ability to provide highly detailed and accurate results. Traditional methods of stress analysis often rely on simplifications or approximations that may not fully capture the behavior of complex components like connecting rods under dynamic loads. FEM, however, allows for precise modeling of stress distribution throughout the entire connecting rod, including areas that are otherwise difficult to analyze using conventional methods. Another key advantage is that FEM can simulate real-world conditions, including dynamic loading, temperature variations, and the effects of material nonlinearity. This makes it possible to optimize the connecting rod for conditions that it will experience in actual engine operation, ensuring that the component performs well under all circumstances. Furthermore, FEM can help optimize designs for various types of engines, whether they are high-performance racing engines or smaller, low-power engines, providing valuable insights into how different engine specifications affect the performance of the connecting rod. FEM also reduces the need for costly and time-consuming physical testing. Instead of building and testing multiple prototypes, engineers can evaluate hundreds of potential designs virtually, saving both time and money.

Additionally, FEM allows for the optimization of not only the stress distribution but also other factors such as material usage and cost, which can be crucial in designing cost-effective components that still meet performance and durability requirements.

4. Applications in Connecting Rod Design

The application of the Finite Element Method (FEM) in the design of connecting rods is crucial for optimizing their performance, durability, and weight efficiency. Connecting rods are subjected to high dynamic loads and stresses due to the reciprocating motion of the piston in internal combustion engines. These forces vary depending on the operating conditions, and understanding how the connecting rod responds to such loads is key to improving engine efficiency and reducing failure risks. FEM plays a significant role in this process by allowing engineers to simulate the performance of connecting rods under various conditions and optimize

their design to withstand stresses without failure. FEM provides a detailed, numerical approach to stress analysis. By breaking down the complex geometry of the connecting rod into smaller elements, FEM can simulate how the component behaves under real-world loads.

This analysis allows engineers to identify critical areas of stress concentration and assess how changes in the geometry, material properties, and design could improve the component's performance and longevity. For instance, FEM can help design connecting rods that are both strong and lightweight, improving the overall efficiency of the engine while maintaining reliability under stress. In practical applications, FEM has been used to optimize the design of connecting rods in various engine types, from automotive engines to smaller engines used in two-wheelers. The ability to model and simulate stress distributions across the entire component helps engineers make informed decisions about material selection and design modifications that can enhance the performance and durability of the connecting rod.

- **Case Studies of FEM in Connecting Rod Design**

Numerous studies have demonstrated the value of FEM in connecting rod design, showcasing its ability to optimize strength, weight, and fatigue resistance. Singh et al. (2017) performed FEM-based optimization on connecting rods in automotive engines, focusing on reducing stress concentrations by adjusting the geometry and material distribution. Their study found that FEM simulations could identify areas of excessive stress, enabling design modifications that significantly improved the component's durability.

By refining the geometry and modifying the material properties of critical areas, the engineers were able to extend the connecting rod's lifespan and enhance its fatigue resistance. Similarly, Gurunath et al. (2015) used FEM to analyze the stress distribution in connecting rods subjected to dynamic loads. The study focused on understanding how the big end of the connecting rod, a known high-stress zone, could be redesigned to reduce fatigue failure. The results of their FEM simulations revealed that altering the geometry and introducing features like fillets in the high-stress areas could significantly enhance the rod's performance, reducing the likelihood of failure under cyclic loading.

This case study demonstrated the critical role that FEM plays in optimizing designs for dynamic loading conditions and improving the overall reliability of connecting rods. Swapnil and Saifullah (2014) applied FEM to the design of a connecting rod for a two-wheeler engine. Their study explored how different geometries and material properties could affect the stress distribution and performance of the connecting rod. The use of FEM simulations enabled the engineers to identify the best design configuration that balanced strength and weight while ensuring durability under engine stresses. This research further extended the application of FEM to smaller engine designs, proving that FEM can be applied across a wide range of engine types for optimal connecting rod performance.

- **Optimizing Material Selection with FEM**

FEM not only helps optimize the geometry of the connecting rod but also plays a significant role in material selection. The choice of material impacts the fatigue resistance, strength, and overall performance of the connecting rod. FEM allows for the simulation of how different materials will behave under stress and fatigue, enabling engineers to select the most appropriate material for the intended application. In traditional designs, connecting rods are often made from steel, aluminum, or a combination of alloys that offer a balance of strength and weight. However, with the increasing demand for high-performance engines, advanced materials such as titanium alloys and composites are being considered for their superior strength-to-weight ratio. FEM simulations can model the performance of these advanced materials, highlighting how they respond to dynamic loads and how they can be utilized to improve the overall performance of the connecting rod. By using FEM to analyze the behavior of various materials under engine conditions, engineers can make informed decisions about material selection, ensuring that the connecting rod is optimized for strength, weight, and fatigue resistance.

Furthermore, FEM can simulate the material's performance over extended periods, predicting how it will degrade under cyclic loading and environmental conditions. This capability ensures that the selected material will perform effectively throughout the life of the engine, reducing maintenance costs and increasing engine reliability.

- **Performance and Durability Enhancement with FEM**

The impact of FEM on the performance and durability of connecting rods is profound. FEM allows engineers to evaluate how connecting rods will perform under a range of loading conditions, ensuring that they can withstand high stresses and loads over extended periods without failure. This is especially critical in high-performance and racing engines, where durability is essential for maintaining engine reliability under extreme operating conditions. FEM can also predict the fatigue life of connecting rods, which is crucial for understanding how the component will perform over time. Fatigue failure, caused by repeated loading cycles, is a common issue in connecting rods, and FEM simulations can help predict the number of cycles a connecting rod can endure before failure occurs. By identifying potential weak spots and optimizing the design for fatigue resistance, FEM can significantly enhance the lifespan of the connecting rod and reduce the likelihood of engine failure. Moreover, FEM allows for optimization not only of the strength but also of the weight of the connecting rod. Weight reduction is particularly important in high-performance engines, where reducing weight can lead to improvements in overall engine efficiency and performance. By optimizing the geometry and material distribution within the connecting rod, FEM ensures that the component is as lightweight as possible while still maintaining its strength and durability. This is a key factor in improving fuel efficiency and overall engine performance.

5. Challenges and Limitations

- **Accuracy of Material Models:** Real-world materials exhibit non-linear behavior under stress, such as plastic deformation or fatigue, which can be difficult to model accurately in FEM. Inaccurate material models can lead to unreliable predictions, affecting the optimization process and ultimately the component's performance.
- **Complex Boundary Conditions:** Setting up accurate boundary conditions that reflect real engine conditions is challenging. The connecting rod experiences dynamic loads, temperature changes, and varying forces, and modeling these accurately requires extensive data, which may not always be available or easy to implement.
- **Computational Intensity:** FEM simulations, especially for complex geometries or when considering multiple physical phenomena (multiphysics), require significant computational power. This can result in long simulation times and necessitate the use of high-performance computing resources, which may not be feasible for all design teams.
- **Mesh Quality and Resolution:** The accuracy of FEM results is highly dependent on mesh quality. A fine mesh increases accuracy but also computational costs, while a coarse mesh may miss critical stress concentrations, leading to potential design flaws or inefficiencies in the optimization process.
- **Validation of Simulation Results:** FEM-based simulations need experimental validation to confirm their reliability. Without physical testing or sufficient real-world data, the results of FEM analysis may not fully align with actual component performance, limiting the effectiveness of the design optimizations derived from simulations.

6. Conclusion

The Finite Element Method (FEM) has proven to be a vital tool in the design and optimization of connecting rods, offering engineers a detailed and accurate approach to stress analysis and performance optimization. By dividing a complex component like a connecting rod into smaller, manageable elements, FEM allows for the simulation of how it behaves under various operational conditions, such as dynamic loading, thermal effects, and cyclic stresses. This capability enables engineers to identify critical areas where stress concentrations occur and optimize the design to improve the component's fatigue resistance, strength, and overall durability.

FEM's ability to simulate real-world conditions reduces the need for costly and time-consuming physical prototypes, making it a highly efficient and cost-effective method for optimizing connecting rod designs. Additionally, FEM facilitates the selection of optimal materials and geometries by allowing engineers to test different configurations virtually. This leads to lighter, stronger components that improve engine efficiency and reduce the likelihood of failure under operational stresses. However, despite its many advantages, the use of FEM comes with challenges. The accuracy of material models, setting realistic boundary conditions, and the computational intensity of running complex simulations are some of the limitations that need to be addressed.

Furthermore, while FEM provides valuable insights into the behavior of connecting rods, its results must be validated through experimental testing to ensure they align with real-world performance. In conclusion, FEM is an indispensable tool in modern mechanical design, particularly in optimizing connecting rods. As computational power increases and advancements in material modeling and multiphysics simulations continue, FEM will likely become even more effective and efficient, offering new possibilities for designing high-performance, durable engine components. Despite the challenges, the continued evolution of FEM will further cement its role in advancing engine technology.

References

1. Singh, V., Sumit, V., Harish, R. C., Vishal, B., & Abhinesh, B. (2017). Stress optimization of connecting rod using finite element analysis. *International Journal of Research in Applied Science & Engineering Technology (IJRASET)*, 5(6), 1152-1178.
2. Gurunath, S., Vahid, J., Ganesh, S., Jadhav, V., & Vinayak, D. Y. (2015). Stress analysis of connecting rod using FEA. *ICRAME2015 (GH Raisonny COE, Pune)*.
3. Swapnil, B. I., & Saifullah, K. (2014). Modelling and analysis of connecting rod of two wheeler (Hero Honda Splendor). *International Journal of Analytical, Experimental and Finite Element Analysis (IJAEFEA)*, 3(2), 38-41.
4. Mohammed, M. A., & Mohamed, H. (2015). Analysis of fatigue stresses on connecting rod subjected to concentrated loads at the big end. *4th International Conference on Materials Processing and Characterization, India*.
5. Moon, K. L., Hyungyi, L., Tae, S., & Hoon, J. (2010). Finite element analysis of connecting rods. *Journal of Materials & Design*, 31(6), 2796-2803.
6. Adnan, A. H., Akash, K., Ajinkya, C., Moin, K., & Suresh, P. (2018). Stress optimization of connecting rod using FEM. *International Research Journal of Engineering and Technology (IRJET)*, 5(4), 282-285.
7. Ramani, J., Sunil, S., & Pushendra, S. (2014). Stress analysis of connecting rod using FEA. *International Journal of Engineering Research and Applications*, 3(4), 216-220.
8. Suherman, S., Bambang, W., Achmad, P., Widodo, B. S., & Arifin, N. (2016). Modeling and static analysis of a connecting rod in range extender engine. *International Conference on Sustainable Energy Engineering and Application (ICSEEA), Jakarta, Indonesia*.
9. Savanoor, R., Abhishek, P., Rakesh, P., & Amit, R. (2014). Stress optimization of connecting rods. *International Journal of Mechanical Engineering & Robotic Research*, 3(5), 511-516.
10. NPTEL Course Web IIT. (2009). Module 1: Introduction to finite element analysis. *Indian Institute of Technology*.
11. Roylance, D. (2011). *Finite element analysis*. Massachusetts Institute of Technology.
12. Shinde, V., & Gurunath, V. (2015). *Fundamental concepts of finite element method*. DACOE, Karad, India.
13. Nikishkov, G. (2004). *Introduction to the finite element method*. University of Aizu, Aizu-Wakamatsu, Japan.

14. Dixit, U., & S, U. (2014). Finite element method: An introduction. Department of Mechanical Engineering, Indian Institute of Technology, Guwahati, India.
15. Bathe, K. J. (2014). Finite element procedures. Prentice Hall, Pearson Education, Inc.
16. ANSYS. (2017). ANSYS Fluent tutorial guide. ANSYS, Inc.
17. Erdogan, A., & Ibrahim, G. (2015). The finite element method and applications in engineering using ANSYS. Springer International Publishing.
18. Nakasone, Y., Yoshimoto, S., & Stolarski, T. (2006). Engineering analysis with ANSYS software. MPG Books Ltd., Bodmin, Cornwall, UK.
19. ANSYS. (2019). Introduction to a finite element analysis program: ANSYS. Retrieved from http://innomet.ttu.ee/martin/mer0070/wb/ws1/ansys_classic_tutorial_lab1_.pdf
20. Wang, G., Guoqing, Z., Yang, J., Guan, Y., Gong, W., & Zhang, H. (2012). Modal analysis of high-speed spindle based on ANSYS. 7th International Conference on Computer Science & Education (ICCSE), Melbourne, VIC, Australia.
21. Kim, B. S., Lee, S. H., Lee, M. G., Ni, J., Song, J., & Lee, W. (2017). Stress analysis and optimization using finite element method. Journal of Materials Processing Technology, 187-188, 30-36.
22. Jang, G., Lee, S. H., & Jung, M. S. (2002). Vibration analysis using finite element method. Journal of Sound and Vibration, 251(1), 59-78.
23. Sayeed, A. G., Sirajuddin, E. K., & Syed, H. S. (2014). Stress optimization and fatigue analysis of connecting rods. International Journal of Innovative Research in Science, 3(4), 16674-16687.
24. Barbero, E. J. (2014). Finite element analysis of composite materials using ANSYS. CRC Press, Taylor and Francis Group.
25. Muhammad, A., & Shanono, I. H. (2018). Stress analysis of connecting rods. Nigerian Journal of Technological Research, 13(2), 14-19.

