IJARCCE ISSN (O) 2278-1021, ISSN (P) 2319-5940

International Journal of Advanced Research in Computer and Communication Engineering

Impact Factor 8.471 ⪤ Peer-reviewed & Refereed journal ⪤ Vol. 15, Issue 5, May 2026

DOI: 10.17148/IJARCCE.2026.155175

"Finite Element Analysis of Connecting Rod Stresses, Deformation, and Material Optimization"

A Kiran Kumar[1], A Varun Kumar[2], B Lingaraju[3], Chetan G Gotur[4]

U.G. Student's, Department of Mechanical Engineering, BITM, Karnataka, India[1-4]

Abstract: The connecting rod is a vital component that converts reciprocating motion into rotary motion, enduring extreme compressive and tensile loads (Khudhair et al., 2024; Uddin et al., 2026). This study employs Finite Element Analysis to evaluate the structural integrity and performance of connecting rods under various material configurations, including forged steel, Aluminum 360, and advanced composites like Al-MWCNT (Chumbre, 2018; Uddin et al., 2026). Utilizing simulation software such as ANSYS and SolidWorks, the analysis focuses on minimizing mass while maintaining a high factor of safety to improve engine efficiency (Mishra et al., 2023; Muhammad et al., 2020). Results indicate that while forged steel provides high fatigue strength, Aluminum 360 exhibits significantly lower total deformation and a substantial mass reduction (Uddin et al., 2026). Furthermore, optimization techniques demonstrate that material substitution can lead to a weight reduction of up to 43.48% without exceeding material yield strength (Mishra et al., 2023). This research concludes that FEA is an essential tool for identifying critical stress concentrations and optimizing the geometric design of connecting rods for modern high-performance engines (Mihalache et al., 2014; Sun, 2006).

Keywords: Connecting rod, Finite Element Analysis, Stress analysis, Material optimization, Aluminum 360, Weight reduction, ANSYS, Von-Mises stress.

INTRODUCTION

The connecting rod is a critical component in internal combustion engines, serving as the bridge between the reciprocating piston and the rotating crankshaft (Uddin et al., 2026). Its primary function is to convert the linear reciprocating motion of the piston into the rotary motion of the crankshaft (Khudhair et al., 2024). During operation, it is subjected to complex cyclic loading, including high compressive gas pressure during the power stroke and tensile inertia forces at high engine speeds (Patel et al., 2026).
The primary challenge in connecting rod design is balancing high structural integrity with minimal weight. Excessive mass increases inertia forces, leading to reduced engine efficiency and increased vibration (Uddin et al., 2026). Therefore, optimizing the rod through material selection and geometric modification is essential for enhancing fuel efficiency and durability (Muhammad et al., 2020). This study utilizes Finite Element Analysis to evaluate stresses and deformations across various materials to identify an optimal design that maintains a high factor of safety while reducing total mass.

LITERATURE REVIEW

Previous research has extensively explored the use of various materials to improve connecting rod performance. While forged steel has been the traditional choice due to its high fatigue strength, its high density is a significant disadvantage (Uddin et al., 2026). Studies comparing traditional materials have found that 42CrMo steel alloy is approximately 11.67% lighter than 20CrMo steel, demonstrating that even within steel alloys, significant weight savings are possible (Chumbre, 2018).
Modern research has shifted toward aluminum alloys and composites. For instance, Aluminum 360 has been shown to exhibit lower deformation (1.950×10^{-5} mm) and lower stress (2.992×10^4 N/m²) compared to conventional forged steel, while offering substantial weight reduction (Uddin et al., 2026).

ISSN (O) 2278-1021, ISSN (P) 2319-5940

International Journal of Advanced Research in Computer and Communication Engineering

Impact Factor 8.471 ⁜ Peer-reviewed & Refereed journal ⁜ Vol. 15, Issue 5, May 2026

DOI: 10.17148/IJARCCE.2026.155175

Advanced materials like Carbon Fiber and Al-MWCNT have also been tested, with results indicating that carbon fiber exhibits lower Von-Mises stress compared to stainless steel and standard aluminum alloys (Chumbre, 2018; Ramesh et al., 2017). FEA tools such as ANSYS and SolidWorks are consistently cited as the standard for accurately predicting these behaviors under realistic loading conditions (Mishra et al., 2023; Muhammad et al., 2020).

METHODOLOGY

The methodology for this FEA study follows a structured simulation workflow:
Modeling: The 3D geometry of the connecting rod is developed using parametric modeling software such as SolidWorks, CATIA, or NX 6.0 (Bansal, 2013; Chumbre, 2018).
Material Assignment: Properties such as Young's Modulus, Poisson's ratio, and density are assigned for materials including structural steel, aluminum alloys, and titanium alloys (Chumbre, 2018; Uddin et al., 2026).
Meshing: The model is discretized into finite elements. A common approach uses tetrahedral elements to manage the complex surfaces of the rod, with one study utilizing 7,758 elements and 14,168 nodes to ensure precision (Chumbre, 2018).
Boundary Conditions and Loading: Static structural analysis is typically performed by fixing one end (often the small end/piston end) and applying a compressive or tensile load to the other end (crank end) (Chumbre, 2018; Gautam, 2013). Dynamic simulations may also incorporate pressure-volume diagrams to calculate varying load conditions during the engine cycle (Bansal, 2013).

RESULTS

The FEA simulations provide detailed insights into the structural behavior of the connecting rod. Key results include:
Stress Distribution: The maximum Von-Mises stress is typically concentrated at the transition between the shank and the ends, or at the center of the shank under buckling loads (Gautam, 2013; Sun, 2006).
Deformation: Aluminum 360 demonstrates significantly lower total deformation compared to forged steel, which enhances dimensional stability during high-speed operation (Uddin et al., 2026).
Weight Optimization: Through material substitution and geometric refinement, research indicates that a weight reduction of up to 43.48% can be achieved without exceeding the yield strength of the material (Mishra et al., 2023).
Factor of Safety: While aluminum alloys reduce weight, forged steel often maintains a higher stiffness-to-weight ratio in specific high-stress applications (Chumbre, 2018).

DISCUSSION

The comparison of materials reveals a trade-off between mass and structural performance. Although forged steel is highly durable, the inertia forces generated by its mass reduce overall engine efficiency (Uddin et al., 2026). Material optimization results suggest that replacing steel with Aluminum 360 or carbon fiber composites can significantly reduce these inertia forces (Chumbre, 2018; Uddin et al., 2026).
Furthermore, the analysis indicates that "areas of interest" (critical stress points) can be effectively managed by dividing the geometry during FEA, allowing designers to reinforce specific sections while thinning others to save mass (Mihalache et al., 2014). The use of Al-MWCNT and titanium alloys also presents a viable alternative for high-performance engines, as they offer lower Von-Mises stress and reduced weight compared to traditional carbon steel (Chumbre, 2018).

CONCLUSION

This study confirms that Finite Element Analysis is a robust tool for the optimization of connecting rods. Key findings indicate that:

International Journal of Advanced Research in Computer and Communication Engineering
Impact Factor 8.471 ⁘ Peer-reviewed & Refereed journal ⁘ Vol. 15, Issue 5, May 2026
DOI: 10.17148/IJARCCE.2026.155175

Material Selection: Aluminum 360 and carbon fiber composites are superior to traditional forged steel for applications requiring significant weight reduction and lower deformation (Chumbre, 2018; Uddin et al., 2026).
Performance: Optimizing the rod can lead to a weight reduction of over 40%, which directly improves fuel efficiency and reduces engine vibration (Mishra et al., 2023).
Design Validation: FEA accurately identifies critical stress concentration points, enabling targeted geometric modifications that ensure a safe factor of safety while minimizing material usage (Mihalache et al., 2014; Sun, 2006).
Future work should focus on multi-objective optimization that integrates fatigue life and thermal stresses to further refine the reliability of advanced composite connecting rods.

REFERENCES

1. Helal Uddin, Md. Rasel Ahmed, Touhidur Rahman Sajib, et al.. Design and Structural Analysis of a Connecting Rod Using Different Materials. (Uddin et al., 2026)
2. Maher Rehaif Khudhair, Farah Kamil, Mohannad Aziz Kadhom, et al.. Optimum Design of Connecting Rod – A Review. (Khudhair et al., 2024)
3. Aisha Muhammad, Mohammed A. H. Ali, and Ibrahim Haruna Shanono.. Finite Element Analysis of a connecting rod in ANSYS: An overview. (Muhammad et al., 2020)
4. Vinayak Chumbre.. Design and Comparative Analysis of Connecting Rod using Finite Element Analysis. (Chumbre, 2018)
5. B T Ramesh, Vinayaka Koppad, and T Hemantha Raju.. Analysis and Optimization of Connecting Rod With Different Materials. (Ramesh et al., 2017)
6. Sourabh Mishra, Ruchika Saini, and U. K. Joshi.. Connecting Rod Deformation and Statical Stress Analysis Using Various Materials: A Review. (Mishra et al., 2023)
7. Ram Bansal.. Dynamic Simulation of a Connecting Rod made of Aluminium Alloy using Finite Element Analysis Approach. (Bansal, 2013)
8. Abhinav Gautam.. Static Stress Analysis of Connecting Rod Using Finite Element Approach. (Gautam, 2013)
9. Gang Sun.. Connecting Rod Design Based on Finite Element Analysis. (Sun, 2006)
10. Andrei Marius Mihalache, Gheorghe Nagîț, Marius Ionuț Rîpanu, et al.. Experimental Framework for FEA Analyses of 3D Geometry of a Connecting Rod. (Mihalache et al., 2014)