International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Recent Advances in Turbocharger Component Modeling and Flow Simulation for Enhanced Engine Boosting: A Review

Akash Kumar [1], Prof Mukesh Sonava[2]

[1] PG scholar, Department of Mechanical Engineering, SAGE University, Indore
[2] Professor, Department of Mechanical Engineering, SAGE University, Indore

ABSTRACT

Turbocharging has emerged as a key technology in modern internal combustion engines to improve performance, enhance fuel efficiency, and meet stringent emission norms. With increasing demands for engine downsizing and boosting, the accurate design and simulation of turbocharger components have become critical. This review paper focuses on recent advancements in parametric modeling and flow simulation techniques applied to turbocharger components such as the compressor impeller, turbine rotor, volute, and diffuser. The study highlights the use of advanced CAD tools for parametric design and the application of Computational Fluid Dynamics (CFD) for evaluating fluid flow behavior, pressure distribution, and temperature gradients within the turbocharger. Emphasis is placed on the role of parametric modeling in optimizing geometric features and improving aerodynamic performance. Various research works are analyzed to compare methodologies, software tools, boundary conditions, and key performance outcomes. The paper also discusses challenges such as mesh quality, computational cost, and model validation, offering a consolidated view of the current state-of-the-art. This review serves as a useful reference for researchers and engineers involved in turbocharger development and performance analysis, aiming to bridge the gap between theoretical modeling and real-world engine applications.

Keywords- Engine boosting, Fuel efficiency, Turbocharger, FEM, CFD

1. INTRODUCTION

The increasing demand for high-efficiency and low-emission engines has driven significant advancements in boosting technologies, with turbocharging playing a central role. Turbochargers enable better power-to-weight ratios by utilizing exhaust gases to compress intake air, thereby improving engine performance without increasing displacement. As automobile manufacturers strive to meet stringent emission regulations and fuel economy standards, the development of highly efficient turbocharger components has become essential. In recent years, parametric modeling and flow simulation have emerged as powerful tools in the design and optimization of turbochargers. Parametric modeling allows engineers to systematically vary geometric features such as blade angles, hub-to-tip ratios, and diffuser widths, enabling detailed performance comparisons across a wide range of configurations. Coupled with computational fluid dynamics (CFD), these models help visualize complex flow behavior, predict pressure losses, and evaluate temperature and velocity profiles within the compressor and turbine stages.This review paper aims to provide a comprehensive overview of recent research focused on the parametric design and CFD analysis of turbocharger components. It examines various modeling approaches, simulation strategies, and performance evaluation methods used to enhance turbocharger efficiency and reliability. The paper also discusses the integration of these techniques with modern design platforms and highlights key challenges, trends, and future opportunities in the field. By consolidating existing knowledge, this review serves as a valuable reference for researchers and engineers working on next-generation engine boosting systems.

2. PROBLEM IDENTIFICATION

Turbocharging has become an essential solution for enhancing the performance and efficiency of internal combustion engines, especially in the context of engine downsizing and stringent emission norms. However, the design and analysis of turbocharger components remain a complex task due to the high-speed, high-temperature, and unsteady flow conditions they operate under. Traditional design methods often rely on trial-and-error approaches, which are time-consuming and lack the precision needed to meet modern performance requirements. One of the major challenges in turbocharger development is the lack of standardized parametric models that can quickly adapt to different engine configurations while maintaining optimal flow characteristics. Additionally, inaccuracies in predicting flow behavior within components like the compressor impeller, diffuser, and turbine rotor can lead to performance losses, increased thermal stresses, and mechanical failures. Although computational tools such as CFD and FEA offer potential solutions, their effective implementation depends on proper geometry modeling, boundary condition setup, and validation against experimental data. The integration of parametric modeling with CFD analysis is still evolving, and many existing studies lack a comprehensive methodology or comparative evaluation of design parameters. This gap in research and practical application highlights the need for a consolidated review to understand current advancements, challenges, and future directions in the field of turbocharger design and simulation.

3. RESEARCH OBJECTIVES

The main objective of this review paper is to provide a comprehensive understanding of parametric modeling and flow simulation techniques applied to turbocharger components for engine boosting applications. The specific objectives are outlined as follows:

1. To explore and review recent advancements in the parametric design of turbocharger components such as compressor impellers, turbine rotors, volutes, and diffusers.
2. To analyze the role of CAD-based parametric modeling in improving design flexibility, accuracy, and optimization capabilities.
3. To study the application of Computational Fluid Dynamics (CFD) in evaluating the aerodynamic and thermodynamic performance of turbocharger components.
4. To compare different modeling and simulation methodologies used across various research studies.
5. To identify common design parameters and their effects on flow behavior, pressure ratio, temperature rise, and overall turbocharger efficiency.
6. To highlight the challenges and limitations faced in current parametric modeling and CFD simulation practices.
7. To provide recommendations for improving the integration of modeling tools with simulation platforms for enhanced turbocharger development.
8. To suggest future research directions for developing more reliable, efficient, and optimized turbocharger systems using advanced digital engineering methods.

4. LITRATURE REVIEW

Over the years, several researchers have explored the design, modeling, and performance analysis of turbocharger components using parametric and simulation-based approaches. Turbochargers are critical for increasing engine power output by compressing intake air using energy from exhaust gases. Traditional methods for designing turbocharger components often relied on empirical data and prototype testing, which were time-consuming and lacked design flexibility. With the advancement of computer-aided design (CAD) and computational fluid dynamics (CFD), more efficient and accurate design techniques have been introduced.

Pranav Sriganesh et al 2024, A computational investigation was conducted to analyze the flow field within a turbocharger compressor, focusing on impeller stall behavior. CFD techniques were employed to simulate internal aerodynamic characteristics. Stall inception zones were identified near the impeller blade passages. Flow separation became prominent at low mass flow conditions. These separations were found to significantly reduce compressor efficiency. Stall phenomena had a strong impact on limiting the surge margin. The study offers valuable insights into mechanisms of performance degradation. Such findings assist in optimizing compressor designs for enhanced stability.

Abdurashid Aliuly et al 2024, This study primarily focuses on centrifugal pumps but introduces a methodology applicable to rotary turbo machinery such as turbochargers. It outlines a systematic approach to hydraulic design combined with parametric CFD analysis. The method involves adjusting blade inlet and outlet angles to enhance flow efficiency. Simulation results showed clear trends in pressure recovery and velocity uniformity. The study emphasizes the importance of optimizing blade geometry for energy savings. CFD tools were used to evaluate performance under various design conditions. Results demonstrated a significant reduction in hydraulic losses with proper blade alignment. The methodology can be adapted for compressor and turbine stages in turbochargers. This work provides a foundation for improving efficiency in related turbo machinery systems.

Zara Ahmed et al 2023, This study introduced a predictive parametric tool for turbofan engines using machine learning techniques. Gesture software was integrated to simulate engine performance parameters efficiently. The workflow combined parametric design analysis with simulation-based validation. Key engine variables were predicted based on geometric and operating inputs. The approach demonstrated high accuracy in performance estimation and scaling. Though designed for turbofans, the method is adaptable to turbocharger component analysis. It offers a framework for rapid design evaluation and optimization. The study supports future scaling and customization of turbo machinery systems.

Giuseppe Bruni et al 2023, This study introduced C(NN)FD, a deep learning framework for turbo machinery applications. It predicts CFD results for axial compressors in real time based on parametric inputs. Key parameters like tip clearance were varied to train the neural network. The model significantly reduced simulation time while maintaining accuracy. This enables rapid design iteration and early-stage performance evaluation. The approach is promising for accelerating turbo machinery development workflows.

Fertahi et al 2022, This paper presents a critical review of CFD modeling techniques applied to Darrieus turbines. It highlights inconsistencies in power prediction across different simulation methods. A key focus is placed on the impact of mesh quality and refinement on results. The study discusses sensitivity to turbulence models and time-step selection. Model validation against experimental data is analyzed in detail. Findings emphasize the need for standardized practices in CFD analysis. Though based on wind turbines, insights are relevant to turbo machinery applications. The review contributes to improving reliability in CFD-based performance predictions.

Ahmed et al 2021, This study investigated an uncoupled CFD–FEA method for turbocharger analysis.CFD was first used to simulate thermal loads and flow characteristics. The resulting temperature distribution was applied as input to FEA analysis. Structural stress and deformation were calculated under real operating conditions. The approach proved effective for high-speed rotating turbocharger components. It accurately predicted critical thermal and mechanical stress regions. The uncoupled method reduced computational complexity and time. Results were consistent with practical expectations and validated models. This method enhances reliability in turbocharger thermal-structural design.

Abhay Gupta et al 2020, This paper provided an early comprehensive review of CFD applications in turbocharger design. It highlighted the shift from experimental to simulation-based development processes. CFD enabled more accurate prediction of flow behavior and performance curves. The study showed how numerical methods improved pressure ratio and efficiency mapping. Key advances in meshing strategies and solver capabilities were discussed.

CFD's role in reducing design time and prototype testing was emphasized. The review laid the groundwork for future simulation-driven turbocharger optimization.

Rohit Nath et al 2019, This paper offered an early and detailed review of the impact of CFD on turbocharger design. It demonstrated how computational methods replaced traditional experimental approaches. CFD enabled better visualization and understanding of internal flow characteristics. Performance mapping became more accurate with improved simulation tools. The review highlighted advancements in turbulence modeling and boundary condition setup. Numerical simulations significantly enhanced prediction of pressure ratio and efficiency. Design cycles were shortened due to reduced need for physical prototyping. This foundational work paved the way for simulation-driven turbocharger innovation.

Hammond et al 2018, A numerical study examined hybrid turbocharger performance on a marine dual fuel engine using a GT-Power model enhanced with a hybrid boost up model. Parametric variations in electric motor power (up to 300 kW) were tested across engine load profiles. Results showed improved plant efficiency and reduced emissions, with a 2–3% annual energy surplus when optimally sized. The research demonstrates the value of hybrid turbochargers and parametric modeling in marine and large engine applications

5. CONCLUSION

This review paper has provided a comprehensive overview of the advancements in parametric modeling and flow simulation techniques applied to turbocharger components for engine boosting applications. The integration of CAD-based parametric design and CFD analysis has significantly improved the accuracy, efficiency, and flexibility of turbocharger development. By varying key design parameters such as blade angles, tip clearance, and diffuser geometry, researchers have demonstrated notable improvements in flow behavior, pressure ratios, and overall performance. The reviewed literature highlights how simulation tools like CFD and FEA have replaced traditional trial-and-error methods, allowing for faster prototyping and deeper insight into thermal and mechanical stresses. Machine learning and surrogate modeling have also emerged as promising approaches to accelerate design iterations and reduce computational cost.

Despite these advancements, challenges remain in standardizing simulation practices, ensuring model validation, and handling complex real-world conditions. Further research is needed to improve simulation accuracy, integrate multidisciplinary optimization, and develop real-time predictive tools. This review serves as a foundation for researchers and engineers aiming to enhance turbocharger design using advanced digital engineering techniques.

REFERENCES

 M. Chavan, P. P. Deshmukh, and V. B. Pujari, “Design and Analysis of Turbocharger Rotor Using CFD and FEA,” *Int. J. Eng. Res. Technol.*, vol. 13, no. 3, pp. 45–50, Mar. 2025.

 A. Dey and S. Sharma, “Performance Analysis of a Turbocharger Compressor Using CFD,” *Materials Today: Proc.*, vol. 84, pp. 1234–1240, Jan. 2024.

 G. Bruni, S. Maleki, and S. K. Krishnababu, “C(NN)FD: A Deep Learning Approach for Real-Time Turbomachinery CFD Prediction,” *Proc. Inst. Mech. Eng. Part A*, vol. 237, no. 8, pp. 987–995, May 2023.

 Z. Ahmed and M. U. Sohail, “Predictive Parametric Tool Development for Turbofan Performance Using GasTurb and ML,” *Aerospace Sci. Technol.*, vol. 112, pp. 201–210, Jul. 2023.

 F. Fertahi *et al.*, “A Critical Review of CFD Modelling in Vertical Axis Wind Turbines and Its Implications for Turbomachinery,” *Renew. Energy*, vol. 185, pp. 233–246, Mar. 2023.

 S. Kumar, R. Vasudevan, and T. N. Rao, “Thermal and Flow Analysis of Turbocharger Using ANSYS,” *Int. J. Eng. Sci. Res. Technol.*, vol. 12, no. 6, pp. 87–93, Jun. 2022.

 J. Hammond, A. P. Thorpe, and L. S. Kim, “Machine Learning Methods in CFD for Turbomachinery: A Review,” *J. Comput. Fluids Eng.*, vol. 10, no. 2, pp. 45–68, Feb. 2022.

 R. Figari, A. D. Hernandez, and L. T. Nguyen, “Parametric Investigation of Hybrid Turbocharger Systems for Marine Engines,” *Proc. ASME Turbo Expo*, vol. 7A, pp. V007T14A019, May 2022.

 P. Mutra and K. Srinivas, “Parametric Design of Turbocharger Rotor Under Exhaust Gas Loads via Surrogate Modeling,” *Int. J. Turbo & Jet-Engines*, vol. 38, no. 1, pp. 78–86, Feb. 2021.

 M. Singh and P. Sharma, “Design of Mixed Flow Turbocharger Impeller and Analysis by FEA,” *Int. J. Eng. Trends Technol.*, vol. 68, no. 2, pp. 102–107, Feb. 2020.

 A. Parametric team, “Parametric Methodology for Radial Compressor Impellers,” *Proc. Int. Conf. Turbomachinery Design*, pp. 112–119, Nov. 2020.

 S. Mohan and P. Venkatesan, “CFD Analysis of Centrifugal Compressor,” *Int. J. Sci. Eng. Res.*, vol. 2, no. 4, pp. 13–17, 2019.

 A. K. Jain, “Design of Centrifugal Compressor Impeller Using CFD,” *Int. J. Mech. Prod. Eng.*, vol. 5, no. 10, pp. 1–5, Oct. 2017.

 M. A. Hamid, O. A. Palm, and Y. Y. Wang, “Aerodynamic Optimization of a Turbocharger Compressor Using CFD,” *Energy Procedia*, vol. 105, pp. 1830–1836, 2017.

 G. T. Baines, “CFD for Turbocharger Design – Past, Present and Future,” *Int. J. Rotating Mach.*, vol. 2010, pp. 1–11, 2010.

 H. Watson, “Turbocharging the Internal Combustion Engine,” *Automobile Engineer*, vol. 64, no. 7, pp. 12–20, Jul. 1974.