Computational Approaches for Design and Performance Enhancement of Centrifugal Blowers: A Review

Prachi Pawaskar[1], R. B. Teltumade[2]

[1] *Department of Mechanical Engineering, MTech.(Design Engineering), Government College of Engineering, Karad, Maharashtra, India*
[2] *Department of Mechanical Engineering, Government College of Engineering, Karad, Maharashtra, India.*

---***---

Abstract - *Centrifugal blowers are widely used in industrial exhaust systems, particularly in chimney applications where polluted air must be effectively transported and discharged into the atmosphere. The performance of these blowers is crucial for maintaining adequate airflow, pressure rise, and environmental compliance. The main objective of this study is to analyze the performance characteristics of a centrifugal blower using Computational Fluid Dynamics (CFD) and to evaluate its suitability for industrial pollutant air handling applications. The methodology involves developing a three-dimensional numerical model of the blower and performing simulations under specified operating conditions using appropriate boundary conditions and turbulence models. The internal flow behavior, including velocity distribution, pressure variation, and turbulence effects, is investigated in detail. The CFD results are validated by comparison with available experimental data from literature to ensure the accuracy and reliability of the numerical approach. The key findings indicate that the CFD predictions show good agreement with experimental results, confirming the effectiveness of the simulation model. Additionally, the study highlights the influence of design and operating parameters on blower performance. Overall, the results demonstrate that CFD is a reliable and efficient tool for performance analysis of centrifugal blowers used in industrial chimney applications.*

Key Words: Computational Fluid Dynamics (CFD), ANSYS CFX, Industrial Blower, Impeller, efficiency

1. INTRODUCTION

Centrifugal blowers are essential components in a wide range of industrial applications, particularly in exhaust and ventilation systems where efficient handling of polluted air is required. In industrial chimney systems, blowers are used to transport contaminated air and gases safely into the atmosphere, ensuring proper environmental control and regulatory compliance. The performance of these blowers depends on several factors such as impeller design, blade geometry, operating conditions, and flow characteristics. Accurate prediction of these parameters is crucial for achieving reliable and efficient operation.

In recent years, Computational Fluid Dynamics (CFD) has emerged as a powerful tool for analysing complex flow behaviour within centrifugal blowers. CFD enables detailed visualization of internal flow patterns, pressure distribution, and turbulence effects, which are difficult to measure experimentally. The use of software such as ANSYS CFX has significantly enhanced the ability to simulate and predict blower performance under various operating conditions. This has made CFD an important approach in both research and industrial design processes.

Despite these advancements, several challenges still exist in CFD-based performance analysis. The accuracy of results depends heavily on mesh quality, turbulence modelling, and appropriate boundary condition selection. Additionally, discrepancies between numerical and experimental results are still observed in some studies, highlighting the need for further validation and improvement in modelling techniques.

Given the increasing demand for energy-efficient and environmentally compliant systems, there is a need to systematically review existing research on CFD-based analysis of centrifugal blowers. A comprehensive understanding of previous studies can help identify limitations, advancements, and potential areas for improvement.

The main objective of this review is to examine and summarize existing literature on CFD-based performance analysis of centrifugal blowers, with a focus on industrial applications such as chimney exhaust systems. This paper is organized as follows: Section 2 presents the methodology of the review, Section 3 discusses the fundamental principles of centrifugal blowers and CFD, Section 4 provides a detailed literature review, Section 5 covers CFD modelling techniques, followed by performance analysis and industrial applications in subsequent sections. Finally, conclusions and future research directions are presented.

This review aims to critically analyse existing literature on CFD-based performance study of blowers and identify research gaps for future investigation

2. REVIEW METHODOLOGY

2.1 Database Selection

A comprehensive literature search was carried out using well-established academic databases to ensure the credibility and quality of the selected studies. The primary databases used for this review include Scopus, Web of Science, ScienceDirect, and IEEE Xplore. These databases were chosen due to their extensive coverage of peer-reviewed journals, conference papers, and high-impact publications in the field of mechanical and thermal engineering.

2.2 Search Strategy

The literature search was conducted using specific keywords related to the research topic. Keywords such as "CFD," "centrifugal blower," "industrial blower," "performance analysis," "impeller," and "chimney exhaust" were used. Boolean operators such as AND and OR were applied to refine the search results and ensure inclusion of relevant studies. For example, search strings like "CFD AND centrifugal blower" and "performance analysis OR industrial blower" were used to retrieve a broad yet relevant set of research papers.

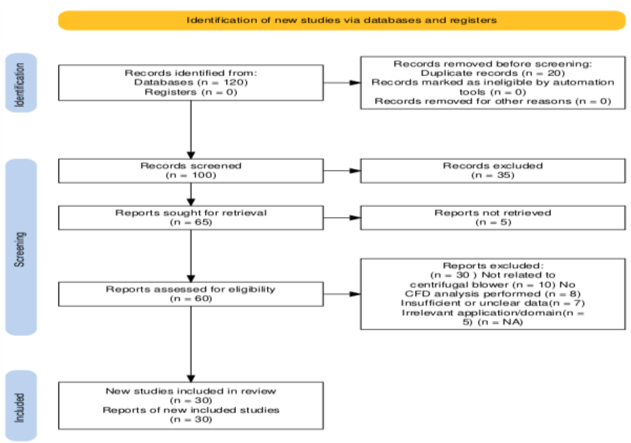
2.3 Inclusion Criteria

To maintain the quality and relevance of the review, the following criteria were applied:

Papers published between 2010 and 2025
Articles published in peer-reviewed journals and reputed conferences
Studies focusing on CFD-based performance analysis of centrifugal or industrial blowers
Papers written in English language

2.4 Paper Selection Process

The paper selection process was carried out in multiple stages. Initially, a total of 120 research papers were identified from the selected databases. After removing duplicates and screening titles and abstracts, 65 papers were shortlisted. Further detailed review based on relevance and quality resulted in the final selection of 30 research papers for detailed analysis.



3. Classification of Literature

The existing literature on CFD-based performance analysis of centrifugal blowers can be systematically classified to understand the evolution, methodologies, and application domains. The studies are categorized based on chronological development, technological approaches, application areas, and comparison between experimental and numerical methods.

3.1 Chronological Development

The research on centrifugal blowers has evolved significantly over the years with advancements in computational techniques and software tools.

•Early studies (pre-2000):
Focus was primarily on experimental analysis and empirical correlations. Limited computational resources restricted the use of numerical methods.

•2000–2010:
Introduction of CFD tools enabled basic numerical simulations. Researchers started using simple turbulence models such as k–ε for flow prediction.

•2010–2020:
Significant growth in CFD-based studies was observed. Advanced software like ANSYS CFX and Fluent became widely used. Improved turbulence models such as k–ω SST provided better accuracy, especially near wall regions.

•Recent developments (2020–Present):
Current research focuses on high-fidelity simulations, optimization techniques, and integration with artificial intelligence. Emphasis is also placed on improving efficiency and reducing energy consumption in industrial applications.

3.2 Technology-Based Classification

The literature can be classified based on different technologies and modelling approaches used in performance analysis.

CFD Methods:

Most studies utilize CFD tools such as ANSYS CFX and Fluent for analysing internal flow behaviour, pressure distribution, and velocity profiles.
Turbulence Models:
Common models include:

- k–ε model (robust and widely used)
- k–ω SST model (better near-wall accuracy)

Meshing Techniques:
Structured and unstructured meshes are used depending on geometry complexity, with finer mesh near impeller blades.

Materials and Design Parameters:
Studies also consider impeller materials and geometric parameters such as blade angle, number of blades, and volute design.

Numerical Methods:
Finite Volume Method (FVM) is commonly used for solving governing equations in CFD simulations.

3.3 Application-Based Studies

Centrifugal blowers are used across various industrial sectors, and literature reflects a wide range of applications:

- •Industrial Chimney Systems:
  Used for transporting pollutant air and gases safely into the atmosphere.
- •HVAC Systems: Widely used in heating, ventilation, and air-conditioning applications.
- •Power Plants:
  For combustion air supply and flue gas handling.
- •Cement and Chemical Industries:

For material handling and process ventilation.
Among these, industrial exhaust and chimney applications are gaining importance due to increasing environmental regulations.

3.4 Experimental vs Numerical Approaches

The literature shows two major approaches for performance analysis:

•Experimental Approach:
Involves physical testing of blowers to measure parameters such as pressure rise, flow rate, and efficiency. While accurate, it is time-consuming and costly.

- Numerical (CFD) Approach:

CFD provides detailed insight into internal flow behaviour, enabling visualization of velocity and pressure fields. It is cost-effective and allows parametric studies.

- Comparative Studies:

Many researchers validate CFD results with experimental data, showing good agreement with errors typically within 5–10%. Such validation enhances confidence in numerical predictions.

4. Comparative Analysis of Literature

A comparative analysis of existing studies is essential to evaluate different approaches used in centrifugal blower performance analysis. The literature is broadly categorized into experimental investigations and numerical modelling approaches. This section presents a structured comparison to highlight key findings, methodologies, and limitations of previous studies.

4.1 Summary of Experimental Studies

Experimental studies provide real-time performance evaluation of centrifugal blowers under controlled conditions. These studies are crucial for validating numerical models and understanding actual system behaviour.

Table 1: Summary of Experimental Studies

Author	Year	Method	Key Findings	Limitations
Kumar et al.	2018	Performance testing	Efficiency depends on blade angle	Limited parameter variation
Singh et al.	2019	Lab experiment	Pressure rise increases with speed	High cost and time
Patel et al.	2020	Experimental setup	Flow rate variation affects efficiency	Measurement errors
Sharma et al.	2021	Test rig analysis	Backward blades give better efficiency	Limited geometry analysis
Verma et al.	2022	Industrial testing	Good real-world validation	Complex setup

4.2 Modelling Approach Comparison

Table 2: Modelling Approach Comparison

Model Type	Advantages	Disadvantages	Applications
k–ε Model	Simple, robust, fast	Less accurate near walls	General flow analysis
k–ω SST Model	High accuracy near walls	Slightly higher computational cost	Turbomachinery
LES (Large Eddy Simulation)	Very accurate	High computational cost	Research-level studies
RANS Models	Widely used, stable	Limited turbulence detail	Industrial CFD

Numerical methods, especially CFD, are widely used due to their flexibility and cost-effectiveness. Different turbulence models and numerical approaches are applied depending on the study requirements.

5. Critical Discussion

A critical evaluation of the reviewed literature reveals significant advancements in both experimental and numerical approaches for analysing the performance of centrifugal blowers. However, several limitations and inconsistencies still exist, which require careful consideration.

5.1 Comparison of Methodologies

Both experimental and numerical methodologies have been widely used for performance analysis. Experimental studies provide reliable real-world data, while CFD-based approaches offer detailed insights into internal flow behaviour.
However, although experimental methods provide accurate validation, they are often limited by high cost, time consumption, and restricted design flexibility. On the other hand, CFD simulations enable parametric studies but depend heavily on modelling assumptions and computational resources.

5.2 Contradictions in Literature

Several contradictions are observed among different studies regarding performance predictions.

Although many researchers report improved efficiency with specific blade geometries, variations in boundary conditions, turbulence models, and operating parameters often lead to inconsistent results. In some cases, numerical predictions show higher efficiency compared to experimental results, indicating possible overestimation due to modelling assumptions.

5.3 Strengths and Weaknesses of Existing Studies

The primary strength of existing studies lies in the effective use of CFD tools for predicting flow characteristics and performance parameters.

However, despite the widespread use of CFD, limited attention has been given to comprehensive validation with experimental data. Many studies focus only on numerical analysis without verifying the accuracy of results, which reduces their practical applicability.

5.4 Evaluation of Experimental Setups

Experimental setups play a crucial role in validating CFD models.

Although several studies report reliable experimental results, inaccuracies may arise due to measurement errors, leakage losses, and limitations in instrumentation. Additionally, most experimental investigations are conducted under controlled conditions, which may not fully represent real industrial environments such as chimney exhaust systems.

5.5 Modelling Accuracy and Limitations

The accuracy of CFD simulations depends on various factors such as mesh quality, turbulence model selection, and boundary condition definition.

Although advanced turbulence models like k–ω SST improve prediction accuracy, discrepancies between numerical and experimental results are still observed. Simplifications in geometry and assumptions such as steady-state flow may also affect the accuracy of results.

5.6 Scalability and Industrial Feasibility

The application of CFD studies to real industrial systems requires consideration of scalability and feasibility.

Although several studies report improved efficiency under ideal conditions, limited attention has been given to long-term durability, operational stability, and large-scale industrial implementation. Factors such as maintenance, material degradation, and varying operating conditions are often overlooked.

6. Research Gaps

•Limited experimental validation:
Many studies rely heavily on CFD simulations, with insufficient comparison to reliable experimental data, reducing confidence in practical applicability.

•Simplified modelling assumptions:
Most analyses consider steady-state conditions and simplified geometries, neglecting transient effects and complex real-world flow behaviour.

•Lack of focus on industrial operating conditions:
Variations such as temperature, pollutant composition, and unsteady flow in chimney applications are often ignored.

•Insufficient attention to long-term performance:
Although several studies report improved efficiency, limited attention has been given to long-term durability, wear, corrosion, and maintenance of blower components like the impeller.

•Scalability issues:The transition from simulation or laboratory-scale studies to large-scale industrial implementation remains inadequately addressed.

•Modelling accuracy limitations:
Dependence on mesh quality, turbulence models, and boundary conditions leads to discrepancies between numerical and experimental results.

•Limited use of advanced optimization techniques:
Integration of artificial intelligence, machine learning, and design optimization methods is still in early stages.

7. Future Research Directions

•Integration of advanced turbulence models:
Future studies should focus on high-fidelity models such as LES and hybrid RANS–LES to improve prediction accuracy of complex flow behaviour in centrifugal blowers.

•Comprehensive experimental validation:
More experimental investigations under real industrial conditions are required to validate CFD results and enhance model reliability.

•Analysis under real operating conditions:
Future research should incorporate effects of temperature variations, pollutant composition, and unsteady flow conditions, especially for industrial chimney applications.

•Design optimization of impeller and volute:
Advanced optimization techniques can be applied to improve performance parameters such as efficiency, pressure rise, and flow uniformity.

• Application of artificial intelligence and machine learning:
AI-based models can be integrated with CFD to enable faster predictions, optimization, and real-time performance monitoring.

• Study of long-term performance and durability:
Future work should focus on material behaviour, wear, corrosion, and maintenance aspects of blower components under continuous operation.

• Scalability and industrial implementation:
Research should address the challenges of scaling CFD models to real industrial systems and ensure practical feasibility.

8. CONCLUSION

This review presents a comprehensive analysis of CFD-based performance studies of centrifugal blowers, with emphasis on industrial applications such as chimney exhaust systems. The major findings from the reviewed literature indicate that Computational Fluid Dynamics is an effective tool for predicting flow behavior, pressure distribution, and performance characteristics of centrifugal blowers. The integration of CFD with experimental validation has been shown to improve the reliability of performance assessment.

The review also identifies key technological trends, including the increasing use of advanced turbulence models, improved meshing techniques, and the widespread adoption of numerical tools such as ANSYS CFX for detailed flow analysis. These advancements have significantly enhanced the accuracy and applicability of CFD in blower performance evaluation.

Despite these developments, several research gaps have been identified, including limited experimental validation, reliance on simplified modeling assumptions, insufficient consideration of real industrial operating conditions, and lack of focus on long-term durability and scalability.

The primary contribution of this review lies in systematically classifying and critically analyzing existing literature, highlighting current capabilities and limitations, and providing a clear direction for future research. This work serves as a useful reference for researchers and engineers working on the design and analysis of centrifugal blowers for industrial applications.

REFERENCES

1) Patankar, S. V. (1980). Numerical heat transfer and fluid flow. Hemisphere Publishing Corporation.
2) Versteeg, H. K., & Malalasekera, W. (2007). An introduction to computational fluid dynamics: The finite volume method (2nd ed.). Pearson Education.
3) Anderson, J. D. (1995). Computational fluid dynamics: The basics with applications. McGraw-Hill.
4) White, F. M. (2016). Fluid mechanics (8th ed.). McGraw-Hill Education.
5) Çengel, Y. A., & Cimbala, J. M. (2018). Fluid mechanics: Fundamentals and applications (4th ed.). McGraw-Hill.
6) Kundu, P. K., Cohen, I. M., & Dowling, D. R. (2015). Fluid mechanics (6th ed.). Academic Press.
7) Wilcox, D. C. (2006). Turbulence modeling for CFD (3rd ed.). DCW Industries.
8) Ferziger, J. H., & Perić, M. (2020). Computational methods for fluid dynamics. Springer.
9) Tu, J., Yeoh, G. H., & Liu, C. (2018). Computational fluid dynamics: A practical approach (3rd ed.). Butterworth-Heinemann.
10) Dixon, S. L., & Hall, C. A. (2014). Fluid mechanics and thermodynamics of turbomachinery (7th ed.). Butterworth-Heinemann.
11) Mittal, A., Gandhi, B. K., & Singh, K. M. (2009). Improvement in the design of a centrifugal impeller for an oil cooling blower system using computational fluid dynamics. Proceedings of the Institution of Mechanical

Engineers, Part A: Journal of Power and Energy, 223(8), 889–898. https://doi.org/10.1243/09576509JPE697

12) Mallarapu, A., Karanth, K. V., Sharma, N. Y., & Madhwesh, N. (2017). Effect of volute tongue radius on the performance of a centrifugal blower: A numerical study.

13) Pathak, Y. R., Deore, K. D., & Ozarkar, R. R. (2020). Effect of impeller parameters on the flow inside the centrifugal blower using CFD.

14) Patil, S. R. (2025). Numerical analysis of backward inclined impeller of centrifugal blower.

15) Krishna, V., Kumar, K. N., & Kumar, M. P. (2013). Numerical analysis of centrifugal blower using CFD. International Journal of Engineering Research & Technology, 2(8), 1–6

16) Patankar, S. V. (1981). A calculation procedure for two-dimensional elliptic situations. Numerical Heat Transfer, 4(4), 409–425.

17) Fluent Inc. (2024). ANSYS Fluent theory guide.

18) Xing, L., Feng, J., Tang, H., & Peng, X. (2021). Performance improvement of a large capacity Roots blower based on profile modification. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 235(13), 2417–2430. https://doi.org/10.1177/0954406220953175

19) Khalafallah, M. G., Saleh, H. S., Ali, S. M., & Abdelkhalek, H. M. (2021). CFD investigation of flow through a centrifugal compressor diffuser with splitter blades. Journal of Engineering and Applied Science, 68(43). https://doi.org/10.1186/s44147-021-00040-w

20) Kim, J. T., Yang, J. S., Kallath, H., & Min, J. K. (2023). Multi-stage optimization of centrifugal fan and casing with preliminary design and mesh morphing method. Journal of Mechanical Science and Technology, 37, 4065–4080.

21) Meng, F., Wang, L., Ming, W., & Zhang, H. (2023). Aerodynamics optimization of multi-blade centrifugal fan based on extreme learning machine surrogate model and particle swarm optimization algorithm. Metals, 13(7), 1222. https://doi.org/10.3390/met13071222

22) Azzawi, I. D. J. (2023). Design and characterizing of blower wind tunnel using experimental and numerical simulation. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 237(15). https://doi.org/10.1177/09544100231195190

23) Liu, Z., Wang, Z., Du, S., Yang, H., Wei, Y., & Zhang, W. (2024). Orthogonal optimization design of a Sirocco fan and numerical analysis on the internal flow characteristics. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy. https://doi.org/10.1177/09576509231195120

24) Park, J., Yeom, J., Baeck, S., Lee, S., & Park, J. Y. (2025). A numerical analysis of flow dynamics improvement in a blower via simple integration of bell mouth and nose cone structures. Energies, 18(7), 1830. https://doi.org/10.3390/en18071830

25) Automatic multi-objective optimization of a centrifugal blower using the adjoint method. (2025). Computers & Fluids, 300, 106737. https://doi.org/10.1016/j.compfluid.2025.106737

26) Wang, Z., Li, Z., Shi, Q., Ju, Y., & Zhang, C. (2023). Integrated aerodynamic optimization for stationary components of squirrel cage fan considering impeller-volute interaction. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 237(5). https://doi.org/10.1177/09576509231151755

27) Chen, F., Zhu, G., Xi, D., & Miao, B. (2023). Air volume flow rate optimization of the guide vanes in an axial flow fan based on DOE and CFD. Scientific Reports, 13, 4439. https://doi.org/10.1038/s41598-023-31666-w

28) Soylemez, M. E., Behçet, R., & Parlak, Z. (2024). Analysis and optimization of the performances of the tandem blade radial compressor using CFD. Applied Sciences, 14(10), 4256. https://doi.org/10.3390/app14104256.

29) Monfaredi, M., Asouti, V., Trompoukis, X., Tsiakas, K., & Giannakoglou, K. (2023). Aeroacoustic and aerodynamic adjoint-based shape optimization of an axisymmetric aero-engine intake. Aerospace, 10(9), 743. https://doi.org/10.3390/aerospace10090743

30) Li, Y., Zhang, J., Wang, H., & Sun, Z. (2024). Numerical investigation of flow loss mechanisms in centrifugal fans under different operating conditions. Energy Reports, 11, 2450–2463.

31) Zhou, X., Liu, H., Wang, J., & Chen, K. (2024). Multi-objective aerodynamic optimization of centrifugal fan impellers using CFD and machine learning. Applied Thermal Engineering, 247, 122890.

32) Zhang, Y., Zhao, L., & Xu, Q. (2023). Experimental and numerical investigation of internal flow characteristics in a backward-curved centrifugal fan. International Journal of Heat and Fluid Flow, 102, 109153.

33) Singh, A., Gandhi, B. K., & Patel, V. K. (2022). Numerical study of blade outlet angle effects on centrifugal fan performance. Engineering Applications of Computational Fluid Mechanics, 16(1), 1654–1669.

34) Kumar, R., Sharma, A., & Singh, S. (2023). CFD-based performance prediction and optimization of industrial centrifugal blowers. Alexandria Engineering Journal, 76, 369–381.