CFD-Based Design Optimization of Axial Flow Fans: A Review

Omkar Jadhav[1], Sujal Shelke[2], Sampada Dravid[3]

[1,2]*Bachelor of Technology,* [3]*Professor, Mechanical Engineering, BRACT's Vishwakarma Institute of Information Technology, Maharashtra, India*

***

Abstract – *Axial flow fans one most used fans compared to the other Radial Fans in various fields such as domestic appliances, commercial and Industrial use-cases. Mostly found in Digital equipment, Air ventilation systems , The parameters for simulations are blade geometry, rotational speed, hub-tip ratio, Temperature, guide vane arrangement .CFD analysis's rise has led to significant optimizations in almost every field due to the MVP of product or Prototyping phase is sped up ,reducing significant time in iterations by bypassing physical testing. This is the review for the advancements in the field of Axial Flow Fans design and innovations by enhancing significant enhance air flow and efficiency also reducing noise levels*

***Key Words*: (Axial Flow Fan, CFD, Optimization, Aerodynamics, Guide Vanes, Noise Reduction, Efficiency**

1. INTRODUCTION

This In Axial flow fans the air movement is parallel to the shaft axis and they are designed to generate high volumetric flow rates with relatively rising low pressure. Due to its advantages discussed above the use of the fans is preferred in compact spaces where there is need of external ventilation or Air conditioning sources. The design of Axial flow fans maintains the flow in one and straight direction which results in the best suitable fans to use in products like ducts, Open ventilation applications, cooling systems.

Fanbao Chen et al.[1] conduct flow rate optimization based on DOE and CFD of guide vanes in axial flow fans. Mustafa Tutar et al[5] perform CFD study for both individual and combined effects of blade stagger angle and the winglets performance of the fans in controlled and uncontrolled environments. Renhui Liu et al [3] performed Finite element analysis of four structural factors of axial fan blade installation angle, number of blades, deflector plate, rotational speed, drawing fan wind pressure and rotational speed cloud diagram, calculation of axial power, by analyzing the distribution of the cloud diagram to design the shape of the fan blade, and derive the change rule of the wind pressure when changing the structure of the fan[3]. Yaming Fan et al.[6] did study that integrates aerodynamic and acoustic experiments with computational fluid dynamics (CFD) simulations to establish predictive models for flow field characteristics and aerodynamic noise in DEC systems. C.Lee et al.[2] did the study to obtain the optimal spanwise distribution of blade angles and chord length through CBD model and created a optimization algorithm which compares the initial model with the optimal conditions and predicts a optimized model. The base for these studies appears to be the gas turbine theory book by Cohen et al [4].

Conventional methods used previously are outdated and time consuming compared to today's CFD, Algorithms, Numerical calculations predictors based on the older data collected from Empirical charts, Laboratory testing and Trial and Errors present already. While conventional methods allowed to build good machines but were limited to constraints such as long research and development time and expensive prototype testing. Available tools today helps engineers evaluate several designs and make a decision help saving a lot of resources.

This paper is the combined study of recent technological developments in the field of design, development of Axial Flow Fans with focus on tools like CFD analysis, aerodynamic optimization, noise reduction, and future intelligent systems.

1.1 Fan Design Method

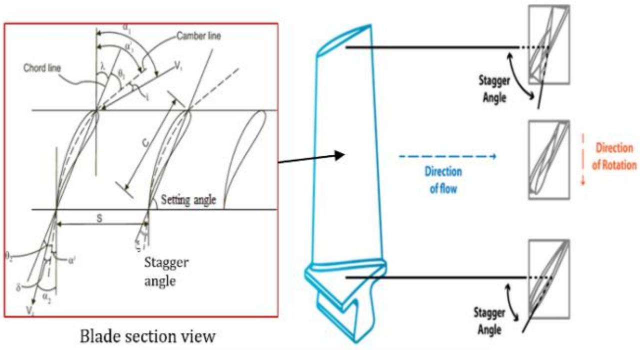

Fig -1: Fan blade section and design parameters[2]

As can be seen in Fig. 1, given the camber angle, setting angle, and chord length, the angle at the leading edge ($\alpha 1'$) and the angle at the trailing edge ($\alpha 2'$) of the blade section are obtained from the following equations

$$\theta c = \alpha 1' - \alpha 2'$$

$$\mu = \pi\ 2 - \xi\ ,\ \tan\xi = \tan \alpha 1' + \tan \alpha 2' / 2 \qquad (2)$$

In this case, the stagger angle is as ξ , and for reference, the setting angle at the hub is the pitch angle of the blade. In addition, the incidence angle (i) at the leading edge of the blade is defined as follows

$$I = \alpha 1 - \alpha 1' \qquad (3)$$

where $\alpha 1$ means the flow angle at the inlet of the blade[]

A 3d model is designed by dividing the bade in 5 sections in the span direction of the fan for CFD analysis and FANDAS .

1.2 Working Principle of Axial Flow fans

In a Axial Flow Fan rotational mechanical energy is converted into airflow energy due to blade design curvature a pressure difference is created from suction to pressure area i.e the motors rotates the impeller and the blade interact with airfoil characteristics with respect to the surrounding environment. The pressure difference generates lift force which changes air momentum increasing pressure rise near the fan. Air flowing through the fan now generates both axial and tangential velocities

Fan performance is representation

- Pressure rise vs flow rate curve
- Efficiency vs flow rate curve
- Power consumption vs flow rate curve
- Noise level vs speed curve

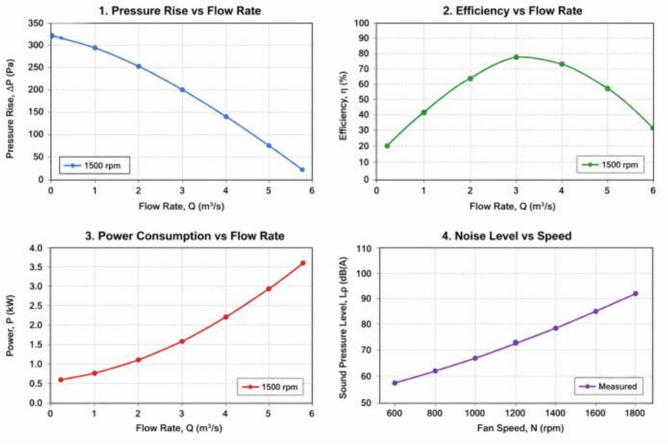

Fig -2: Typical Performance Curves of an Axial Flow Fan

2. Important Design Parameters

2. 1 Blade geometry

Blade geometry is a important factor in designing. Chord length, camber, thickness, pitch angle, stagger angle, and blade twist are the parameters which strongly influence aerodynamic behavior. A good blade design results in smooth flow and suffers minimum losses.

2. 2 Number of Blade

$$Z_b = \frac{6\nu}{(1-\nu)}$$

Increase in blade count improves in pressure difference, But too much blade count increase blockage, friction losses, and Tonal noises. An optimized blade number should be finalized considering above drawbacks.

2. 3 Hub-Tip Ratio

D The Hub-Tip ratio is the ratio of Hub diameter to the overall fan diameter. The hub-tip ratio range of propeller type axial flow fans is less than 0.3 and that of Tube-axial and Vane -axial fan is between 0.3 and 0.5

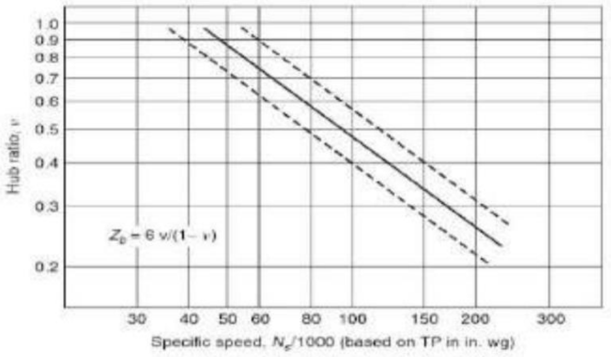

Figure 2.8. The Optimum Hub-Tip Ratio Range for Axial Flow Fans [16]

Fig -3: The optimum hub-tip ratio for Axial flow

2.4 Tip Clearance

The clearance volume between blade tip and casing wall is the Tip Clearance. Leakage flows through this creating vortex and reducing efficiency. Lower tip clearance generally improves performance.

2.4 Rotational Speed

Fan speed strongly affects pressure rise, power consumption, and noise. Higher speed increases airflow but also increases acoustic emissions and energy demand.

3. Fundamental Governing Equations of Axial Flow Fans

Axial flow fan performance is governed by the conservation equations of the fluid flow. In the CFD-based design, these equations are supposedly solved numerically to predict the aerodynamic performance.

3.1 Continuity Equation (Mass Conservation)

For an incompressible flow:

$$\nabla \cdot \vec{V} = 0$$

This ensures that mass is conserved within the flow domain.

3.2 Momentum Equation (Navier–Stokes Equation)

$$\rho\left(\frac{\partial \vec{V}}{\partial t} + (\vec{V}\cdot\nabla)\vec{V}\right) = -\nabla p + \mu\nabla^2\vec{V}$$

Where:

ρ = density
$\vec{V}$ = velocity vector

p = pressure

μ = dynamic viscosity

This equation governs the pressure generation and velocity distribution in axial fans.

3.3 Energy Transfer in Axial Fans (Euler's Equation)

$$\Delta P = \rho \cdot U \cdot (V_{w2} - V_{w1})$$

Where:

U = blade velocity

V_w = whirl component of the velocity

This equation explains how the rotational energy is converted into increase in pressure.

These governing equations are based on standard turbomachinery theory [4]

4. CFD Analysis of Axial flow fans

Computational Fluid Dynamics is one of the important tool present in today's time for fan development. CFD solves governing equations of fluid flow numerically and predicts internal aerodynamic behavior[3] without repeated experiments.

Modern CFD softwares can estimate:

- Velocity contours
- Pressure distribution
- Turbulence intensity
- Wake formation
- Tip leakage vortices
- Flow separation zones
- Overall efficiency

In recent years it is found that that results provided by CFD models are very close to that of experimental results. Common Turbulence models used are as follows :

- Standard k-epsilon model
- RNG k-epsilon model
- K-Omega SST model
- Reynolds Stress Model

Among these the K-Omega SST model is preferred due to its near wall behaviour and separation results are accurate.

Due to CFD Engineers now can test many changes in design like blade angle, blade count, and casing geometry before manufacturing a prototype.

5. Optimization of Blade Performance

Recent research has been focusing on improving fan performance through optimization techniques.

5.1 Blade Angle Optimization

Improper blade angle may cause flow separation and reduced pressure rise. Numerical studies directs to that moderate stagger angles may provide better balance between airflow and efficiency.

5.2 Winglets and Tip Modifications

Winglets attached to near blade tips may help reduce leakage vortices. These features can improve aerodynamic efficiency and lower noise levels.

5.3 Multi-Objective Optimization

Modern designers often optimize more than one parameter , such as:

- Lower power consumption
- Maximum efficiency
- Minimum noise
- Desired pressure rise

Genetic algorithms and response surface methods suggested commonly used [2], [3] for such problems.

5.4 AI – Based design

Machine learning methods are said to being increasingly used to predict performance from historical design data. This reduces computational time compared to repeated CFD runs.

6. Guide Vanes and Flow Recovery

Guide vanes are supposedly said to be stationary blades placed before or after the rotating impeller.

Inlet Guide Vanes

These vanes are there to control the direction of incoming air and improve the incidence angle at the rotor.

Outlet Guide Vanes

These are useful for removing swirl energy and then converting it into useful static pressure.

Studies supposedly shows that vane number, vane angle, spacing, and chord length can significantly affect the final performance. Poor vane design may create additional turbulence, while the optimized vanes may improve pressure recovery and uniformity.

Guide vanes are particularly useful in industrial ventilation systems where the higher static pressure is required.

Guide vanes effects is extensively studied in the literature [1].

7. Aeroacoustics Performance and Noise Reduction

Noise control is said to be becoming a major design requirement, Mostly in HVAC systems, offices, data centres, and household appliances.

Fan noise is mainly generated by:

- Blade passing the frequency tones
- Tip vortex interaction
- Turbulent trailing edge noise

- Flow separation at off-design conditions
- Mechanical imbalance

Proposed methods to reduce the fan noise include:

- Optimized blade spacing
- Serrated trailing edges
- Lower tip clearance
- Smooth blade loading
- Reduced rotational speed
- Better casing design

Recent CFD-acoustic coupling methods are helping identify major sound sources during [6]early design stages.

8. Case Study

8.1: Guide Vane Optimization Using CFD and DOE

A recent study done by Fanbao Chen et al.[1] applied the combined CFD and DOE to optimize the guide-vane parameters in an axial flow fan. The results that the vane chord length had, has the largest significant influence on the airflow simulations, The vane number had a secondary effect But by reducing the vane chord length by 38 mm and changing vane number to 18, the system had said to achieved an optimized flow rate of 142.07 m^3/h at 5000 rpm. This CFD model was then validated against the experimental measurements, revealing the close agreement with less than 5% deviation. This shows that the CFD-driven optimization can really decrease the need for physical prototypes, testing and accelerating the design process

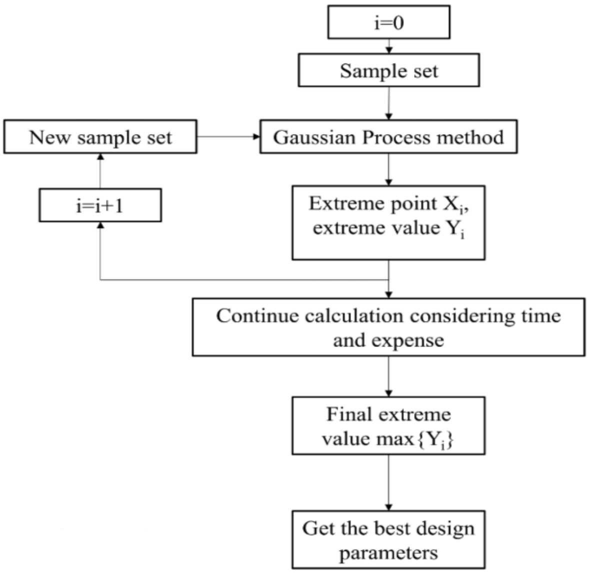

Fig -3: Extreme point search method for Guide vane number[1]

8.2 CFD-Based Blade Optimization and Efficiency Enhancement (2024)

A CFD optimization study done Renhui Liu et al.[3] investigated the influence of these structural parameters such as blade installation angle, number of blades, deflector configuration, and rotational speed on the aerodynamic performance of an axial flow fan. This study has utilized the numerical simulations to generate the pressure-flow (P–Q) characteristics and analyse velocity and pressure distributions made inside the fan.

The results had showed that the fan performance is highly reactive to the blade installation angle and the rotational speed. Within the range considered, the optimal design configuration to improve airflow characteristics and overall efficiency was identified. This further demonstrated that the CFD approach is capable of estimating performance parameters such as pressure rise, flow rate, and efficiency without costly experimental measurements.

The optimization workflow, involving finite volume analysis and iterative simulations, virtually evaluated hundreds of designs. The grid independence study showed that the simulation's accuracy was maintained within a very small error band of about 0.5%.

This adds more weight to the reliability of this computational approach. This confirms that a CFD-based design process quickly evaluates a wide range of design variables, drastically reduces the need for physical prototyping, and thus becomes a significant part of contemporary fan design development.

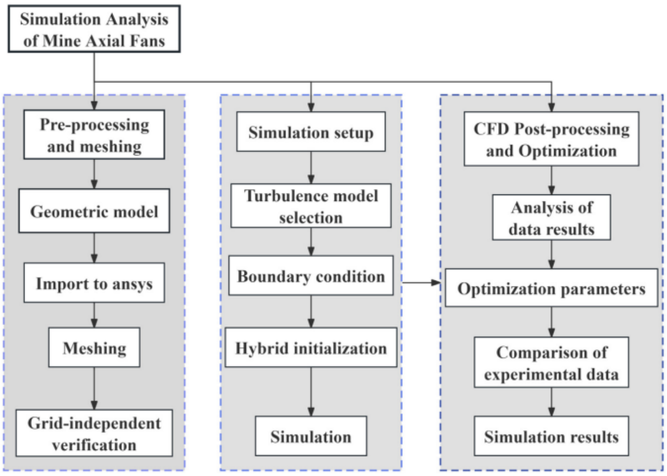

Fig -4: Optimize workflow CFD based simulation[3]

8.3 Hybrid Optimization using Through-Flow Model, CFD and FANDAS (2025)

Recently in 2025, the authors proposed an advanced design methodology that would allow efficient design of high-performance axial fans based on CBD, through-flow modeling, and CFD combined with hybrid optimization algorithms.

A large number of design variables, such as camber angle, setting angle, and chord length distribution along the span, were considered. To automate the exploration of the design space, a metaheuristic algorithm was embedded in the FANDAS design program.

The optimized fan showed an increase in efficiency by about 4.2% compared to the initial design. To verify the model, the CFD results were compared with experimental results. It showed excellent predictive accuracy. Hence, this indicates that the methodology used to estimate the performance of fans before their physical manufacturing is highly reliable. In addition, it has been shown that through-flow modeling combined with CFD is efficient in terms of reducing the computational effort while maintaining high accuracy in this study.

Hence, this hybrid approach will allow more rapid optimization than a fully CFD-based approach and can be handled under multiple design constraints. This study has further demonstrated that using CFD-based optimization algorithms combined with reduced-order models can rapidly design without depending too heavily on physical prototypes.

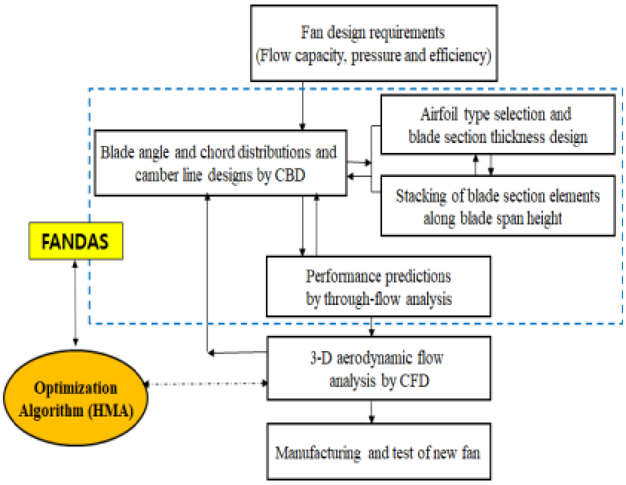

Fig -5: Extreme point search method for Guide vane[2]

8. Applications of Axial Flow Fans

Axial flow fans are the fans widely used across various industrial, commercial, and domestic applications due to their ability to deliver large volumes of air with low pressure and smaller design. Their simple construction, high efficiency, and adaptability makes them the one of the most sought-out for air-moving devices in history of engineering systems.

Industrial Applications

In industrial environments, the axial fans are extensively used for the ventilation, cooling, and exhaust system use-cases. They are said to be commonly installed in the factories, workshops, and process plants to remove heat, fumes, dust, and Toxic gases overtime . In the power plants and the boilers, axial fans can assist in maintaining the airflow for combustion and cooling systems. Cooling towers also supposedly rely heavily on axial fans to enhance the heat dissipation by increasing the airflow across water surfaces.

Infrastructure and Underground Systems

The Axial flow fans plays a critical role in the large-scale infrastructure operations such as tunnels, subways, and mining . In the tunnel ventilation systems, they are supposed to supply fresh air and remove the exhaust gases, especially during all fire emergencies. In the underground mines, axial fans have said to ensured a continuous supply of breathable oxygen and removing the harmful gases and dust particles, improving the worker safety and operational efficiency.

HVAC and Commercial Buildings

In the heating, ventilation, and air-conditioning (HVAC) systems, axial fans are been used in the air handling units, the ventilation ducts, and the exhaust systems. They had helped regulate indoor air quality index by circulating the fresh air and removing all of the stale air. In large scale commercial buildings such as malls, hospitals, and the office complexes, axial fans have contributed to maintaining thermal comfort and proper air distribution.

Electronics and Thermal Management

Small axial fans are widely been used in the electronic cooling applications. Devices like the computers, servers, telecommunications equipment, and power electronics generates significant heat during its operation. Axial fans helps maintain the safe operating temperatures by removing the heat from all sensitive components. In the data centres, high-performance axial fans are been used to ensure efficient airflow management and prevent overheating.

Automotive Applications

In the Automotive sector, axial fans have been used in the radiator cooling systems, and cabin ventilation systems. With the growing adoption of the electric vehicles, axial fans are being used in battery thermal management systems to regulate the temperature and improve battery performance and its lifespan.

Agricultural and Environmental Applications

Axial fans are also being used in the agriculture for the greenhouse ventilation, grain drying, and livestock cooling. Proper air circulation helps maintain the Proper environmental conditions for the crop's growth and animal's health. They are also used in the air pollution control systems to manage the airflow in filtration and exhaust setups.

3. CONCLUSIONS

This paper has reviewed the recent shift in the design of axial flow fans from the empirical, conventional methods to the simulation-based approaches. The Conventional design methods involved multiple iterations of prototyping and experimental testing, which increased the time and cost of development. However, developments in computational fluid dynamics (CFD), design of experiments (DOE), and optimization techniques now allow fans to be virtually tested and optimized before manufacture.

In most recent research, the most important variables affecting the volumetric flow rate, pressure rise, efficiency, and noise generation of axial fans were the blade angle, guide vane shape, and rotational speed. Using a CFD model with an appropriate turbulence model and properly grid-independent checked, one can predict the aerodynamic performance with great accuracy and close agreement with the experimental results.

Indeed, the application of different optimization techniques such as DOE, Gaussian process modeling, gradient-based methods, or hybrid computational frameworks allows design spaces to be explored efficiently. In this way, multiple parameters can be optimized simultaneously, leading to certain improvements in performance and efficiency compared with the initially proposed design and greatly reducing the number of iterations during the design process. Another important result of the studies reviewed is the significant decrease in the need for physical prototyping of the designs. Due to the capabilities of CFD and optimization algorithms, it is possible to analyze several design variants before their realization, thus considerably reducing the number of prototypes and shortening the entire development time. In other words, simulation-driven design reduces costs and increases the reliability and repeatability of the workflow.

Therefore, current designs of axial flow fans rely heavily on integrated computational frameworks that involve CFD simulations, data-driven optimization, and state-of-the-art modeling techniques. Looking into the future, such a process will be enhanced further by introducing artificial intelligence, digital twins, and advanced manufacturing methods to produce low-noise, efficient, and green fans.

REFERENCES

[1] F. Chen, G. Zhu, D. Xi, and B. Miao, “Air volume flow rate optimization of guide vanes in an axial flow fan based on DOE and CFD,” ScientificReports,vol.13,2023.DOI:https://doi.org/10.1038/s41598-023-31666-w.

[2] C. Lee, S. W. Kim, H. T. Byun, and S. H. Yang, “High-efficiency axial flow fan design by combining through-flow modeling, optimization algorithm and computational fluid dynamics simulation,” Journal of Applied Fluid Mechanics, vol. 18, no. 8, pp. 19531963,2025.DOI:https://doi.org/10.47176/jafm.18.8.3325 R. Nicole, “Title of paper with only first word capitalized,” J. Name Stand. Abbrev., in press.

[3] K R. Liu, S. Xu, K. Sun, X. Ju, W. Zhang, W. Wang, X. Ma, Y. Pan,andJ.Li,“CFD analysis and optimization of axial flow fans,”International Journal of Simulation and Multidisciplinary Design Optimization, vol. 15, 2024. DOI: https://doi.org/10.1051/smdo/2024007

[4] H. Cohen, G. F. C. Rogers, and H. I. H. Saravanamuttoo, Gas Turbine Theory, 6th ed., Pearson Education, 2009.

[5] MustafaTutarandJansetBetuCam,“Computational Design of an Energy-Efficient Small Axial-Flow Fan Using StaggeredBladeswithWinglets,”International Journal of Turbomachinery, Propulsion and Power, vol. 10, 2025. DOI: https://doi.org/10.3390/ijtpp10010001

[6] Yaming Fan, Linwei Yu, Jinchai Shen, Yiyu Li, Minfeng Zheng,andShermanC.P.Cheung,“Optimizationofaerodynamic performance and aeroacoustic characteristics in axial flow fans for direct evaporative air conditioning systems,”Engineering Applications of Computational Fluid Mechanics, vol. 19, no. 1, 2025, Article ID: 2569645.DOI:https://doi.org/10.1080/19942060.2025.2569645