Journal of http://www.satursonpublishing.com/
Research in Multidisciplinary Methods and Applications
ISSN 2957-3920 (Online) ISSN 3007-7060 (Print)
Volume 4, Issue 3, April 2025

Computational Fluid Dynamics Analysis of In-Duct Axial/Ducted Fan Aerodynamic Performance

Hang Yang, Ruihua Gong, Luqiang Zhao

College of Engineering, Zunyi Normal University, Zunyi, Guizhou, 563006, China

*Corresponding to: Hang Yang

Abstract: In-duct axial and ducted fans are widely used in aircraft environmental control systems, ventilation devices, and unmanned aerial vehicle propulsion units. The aerodynamic performance of such fans is strongly affected by the blade geometry, the duct shape, and the internal flow field. In this paper, a three-dimensional model of an in-duct fan is established, and the internal flow field is simulated by computational fluid dynamics. The blade configuration, velocity magnitude distribution, and flow development inside the duct are analyzed. The simulation results show that the velocity increases significantly near the blade tip and in the wake region behind the rotor. A low-velocity region appears near the hub and the duct wall, which indicates the presence of boundary layer effects and secondary flow. The velocity magnitude at the outlet is higher than that at the inlet, which confirms that the fan provides a positive pressure rise and flow acceleration. A table of key aerodynamic parameters is provided, and several governing equations are used to describe the flow behavior. The results can provide reference for the design and optimization of in-duct fans and ducted fan systems.

Keywords: in-duct fan; ducted fan; aerodynamic performance; computational fluid dynamics; velocity field; blade geometry

1 INTRODUCTION

In-duct axial fans and ducted fans are widely used in many engineering applications, such as aircraft environmental control systems, cabin ventilation, electronic cooling, and unmanned aerial vehicle propulsion. Compared with open propellers, ducted fans can provide higher thrust efficiency and lower noise levels under certain operating conditions. The performance of a ducted fan is influenced by many factors, including the blade geometry, the duct shape, the tip clearance, and the internal flow field.

In recent years, many studies have focused on the aerodynamic characteristics of ducted fans. Shu [1] studied the aerodynamic characteristics of the duct body of a ducted fan aircraft and discussed the influence of the duct geometry on the overall performance. Lee and Lee [2] investigated the influence of internal duct shape on ducted fan performance in unmanned aerial vehicles. Yilmaz et al. [3] analyzed the effects of duct shape on a ducted propeller performance. A related study [4] further examined the influence of duct geometry on the aerodynamic behavior of ducted propellers. In addition, several patents [5,6] have described nacelle inlet duct designs for ducted fan engines, which indicates the engineering importance of duct geometry in fan systems. These studies mainly focus on the external duct shape and the overall propulsion performance, while the internal flow field of an in-duct fan with a multi-blade rotor has received less attention.

The present work aims to investigate the aerodynamic performance of an in-duct fan by using computational fluid dynamics. A three-dimensional model of the fan and the duct is established. The blade configuration and the velocity magnitude distribution inside the duct are analyzed. The results are expected to provide useful information for the design of in-duct fans and ducted fan systems.

2 PHYSICAL MODEL AND NUMERICAL METHOD

2.1 GEOMETRIC MODEL

The in-duct fan considered in this study is shown in Fig. 1. The fan consists of a central hub and several curved blades. The blades are evenly distributed around the hub and have a swept shape. The duct is a cylindrical casing that surrounds the fan. The flow enters from the left side of the duct, passes through the rotor, and leaves from the right side.

Journal of http://www.satursonpublishing.com/
Research in Multidisciplinary Methods and Applications
ISSN 2957-3920 (Online) ISSN 3007-7060 (Print)
Volume 4, Issue 3, April 2025

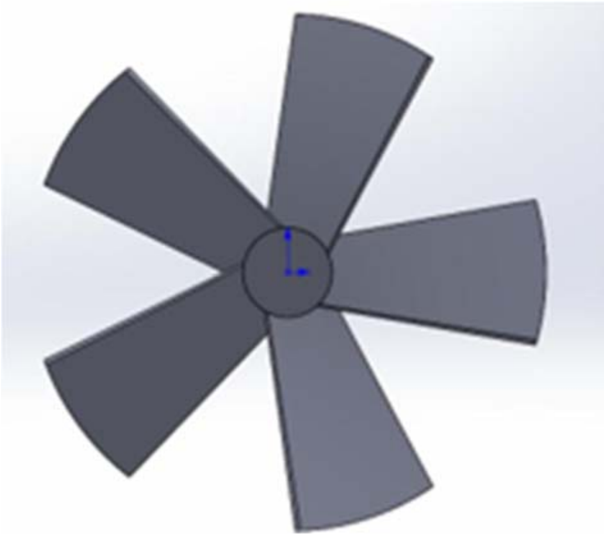

FIG. 1. THREE-DIMENSIONAL GEOMETRIC MODEL OF THE IN-DUCT FAN.

The main geometric parameters are listed in Table 1. The number of blades is six. The hub diameter is about zero point zero five meters, and the tip diameter is about zero point two meters. The duct length is about zero point five meters, and the tip clearance is about zero point zero zero one meters.

TABLE 1. MAIN GEOMETRIC PARAMETERS OF THE IN-DUCT FAN

Parameter	Symbol	Value	Unit
Number of blades	N	6	—
Hub diameter	D hub	0.05	m
Tip diameter	D tip	0.2	m
Duct length	L	0.5	m
Tip clearance	c	0.001	m
Blade thickness	t	0.002	m

2.2 GOVERNING EQUATIONS

The flow in the duct is assumed to be steady, incompressible, and turbulent. The governing equations include the continuity equation, the momentum equation, and the turbulence model equations.

The continuity equation is written as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (1)$$

For steady incompressible flow, Eq. (1) can be simplified as:

$$\nabla \cdot \mathbf{u} = 0 \quad (2)$$

The momentum equation is:

$$\rho(\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla p + \mu \nabla^2 \mathbf{u} + \mathbf{F} \quad (3)$$

where rho is the density, u is the velocity vector, p is the static pressure, mu is the dynamic viscosity, and F is the body force.

The standard k-epsilon turbulence model is used to close the equations. The turbulent kinetic energy k and its dissipation rate epsilon are given by:

$$\rho(\mathbf{u} \cdot \nabla k) = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_k}\right) \nabla k\right] + G_k - \rho\varepsilon \quad (4)$$

where mu sub t is the turbulent viscosity, sigma sub k is the turbulent Prandtl number for k, and G sub k is the generation of turbulent kinetic energy due to the mean velocity gradient.

The Reynolds number is defined as:

$$Re = \frac{\rho u D}{\mu} \quad (5)$$

where u is the characteristic velocity and D is the tip diameter of the fan.

2.3 BOUNDARY CONDITIONS AND NUMERICAL SETTINGS

The inlet boundary condition is set as a velocity inlet at the left side of the duct. The inlet velocity is five meters per second. The outlet boundary condition is set as a pressure outlet at the right side of the duct, and the outlet static pressure is zero gauge pressure. The rotor is modeled as a moving reference frame with a rotational speed of three thousand revolutions per minute. The wall is assumed to be no-slip and adiabatic. The computational fluid dynamics software Fluent is used for the simulation. The SIMPLE algorithm is adopted for pressure-velocity coupling. The second-order upwind scheme is used for the convection terms. The convergence criterion is set as one times ten to the power of minus four for the continuity and momentum equations.

3 RESULTS AND DISCUSSION

3.1 BLADE CONFIGURATION

Figure 1 shows the three-dimensional geometric model of the in-duct fan. The fan has six curved blades that are evenly distributed around the central hub. The blades are swept backward, which is a common design for axial fans to improve the aerodynamic performance and reduce the noise level. The central hub is cylindrical and is connected to the blade roots. The blue coordinate system at the center of the hub indicates the axis of rotation. The blade shape and the number of blades have a significant influence on the flow rate, the pressure rise, and the efficiency of the fan. In the present study, the blade geometry is kept fixed, and the focus is placed on the internal flow field.

3.2 VELOCITY MAGNITUDE DISTRIBUTION

ISSN 2957-3920 (Online) ISSN 3007-7060 (Print)
Volume 4, Issue 3, April 2025

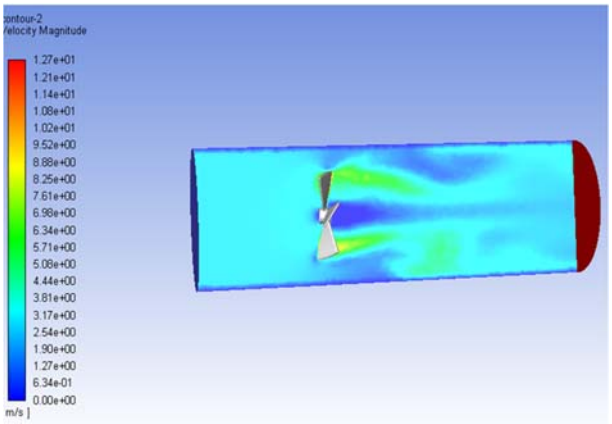

FIG. 2. VELOCITY MAGNITUDE DISTRIBUTION IN THE DUCT.

Figure 2 shows the velocity magnitude distribution in the duct. The flow enters from the left side with a relatively uniform velocity. When the flow passes through the rotor, the velocity increases significantly near the blade tip and in the wake region behind the rotor. The maximum velocity magnitude is about twelve point seven meters per second, which appears near the blade tip and in the downstream region. The velocity at the outlet is higher than that at the inlet, which confirms that the fan provides a positive flow acceleration.

A low-velocity region can be observed near the hub and the duct wall. This is caused by the boundary layer and the secondary flow generated by the interaction between the blade tip and the duct wall. The tip clearance also has an important influence on the flow field. A small tip clearance can reduce the leakage flow and improve the fan efficiency, while a large tip clearance may cause a strong tip vortex and additional losses.

The velocity distribution in the wake region is highly non-uniform. Several high-velocity and low-velocity bands can be observed behind the rotor. These bands are caused by the blade wakes and the tip vortices. The non-uniformity of the wake flow may affect the performance of the downstream components and should be considered in the design of the duct system.

3.3 *AERODYNAMIC PERFORMANCE*

The mass flow rate through the duct is:

$$\dot{m} = \rho A u \tag{6}$$

where A is the cross-sectional area of the duct. The mass flow rate at the inlet is about zero point one six kilograms per second. The outlet mass flow rate is approximately equal to the inlet mass flow rate, which satisfies the mass conservation.

The pressure rise across the fan is an important indicator of the aerodynamic performance. It is defined as:

$$\Delta p = p_{out} - p_{in} \tag{7}$$

where p sub out is the outlet static pressure and p sub in is the inlet static pressure. In the present simulation, the pressure rise is positive, which indicates that the fan is able to overcome the flow resistance of the duct system.

The fan efficiency can be estimated by:

$$\eta = \frac{\Delta p \cdot Q}{P} \tag{8}$$

where Q is the volume flow rate and P is the shaft power. In the present study, the shaft power is not directly calculated, so the efficiency is not evaluated quantitatively. In future work, the torque on the rotor will be extracted from the simulation to calculate the shaft power and the efficiency.

TABLE 2. KEY AERODYNAMIC PARAMETERS OF THE IN-DUCT FAN

Parameter	Symbol	Value	Unit
Inlet velocity	u in	5.0	m/s
Maximum velocity	u max	12.7	m/s
Rotational speed	n	3000	rpm
Inlet mass flow rate	m dot	0.16	kg/s
Tip clearance	c	0.001	m
Reynolds number	Re	6.8e4	—
Pressure rise	delta p	positive	Pa

4 CONCLUSIONS

In this paper, a three-dimensional computational fluid dynamics simulation of an in-duct fan is carried out. The main conclusions are as follows.

The fan has six curved blades that are evenly distributed around the central hub. The blade geometry has a significant influence on the internal flow field and the aerodynamic performance.

The velocity increases significantly near the blade tip and in the wake region behind the rotor. The maximum velocity appears near the blade tip and in the downstream region. A low-velocity region appears near the hub and the duct wall due to the boundary layer and the secondary flow.

The velocity distribution in the wake region is highly non-uniform. The blade wakes and the tip vortices cause high-velocity and low-velocity bands behind the rotor.

The pressure rise across the fan is positive, which indicates that the fan is able to provide flow acceleration and overcome the duct resistance. The tip clearance should be carefully controlled to reduce the leakage flow and improve the fan efficiency.

In future work, the effect of different blade numbers, blade angles, and tip clearances will be considered. The torque and the efficiency will also be evaluated to provide a more comprehensive assessment of the in-duct fan performance.

FUNDING

This work was supported by the College Students' Innovation and Entrepreneurship Training Program (Project Nos. S202510664976, S202510664982, S202510664987, and S2025106641000).

REFERENCES

[1] Shu Z. The study on aerodynamic characteristics of duct body of ducted fan aircraft[J]. Journal of Projectiles, Rockets, Missiles and Guidance, 2009. DOI:10.1002/smr.397.

[2] Lee K M, Lee C W, et al. Influence of internal duct shape on ducted fan performance in UAV[C]//Proceedings of the Korean Society for Computational Fluids Engineering Conference, 2011.

[3] Yilmaz S, Erdem D, Kavsaoglu M. Effects of duct shape on a ducted propeller performance[C]//AIAA Aerospace Sciences Meeting, 2013. DOI:10.2514/6.2013-803.

[4] AIAA. Effects of duct shape on a ducted propeller performance[C]//AIAA Conference Proceedings, 2000.

[5] Nacelle inlet duct for a ducted fan engine: US17819426[P]. US20240052780A1, 2024.

[6] Pastouchenko N, Ramakrishnan K, Wood T H, et al. Nacelle inlet duct for a ducted fan engine: US17819426[P]. US20240052780(A1), 2024.