Research on the Radial Blowing Method for Dynamic Stall Control of Retreating Rotor Blades

LIU Muqi^1, WU Fengfeng^1,2, HALAHAN M^2, LIAO Wenyi^1, LI Yi^1

2026-06-22

ChinaRxiv

Abstract

Blowing control is one of the effective technical approaches for suppressing dynamic stall on the retreating blade of a helicopter rotor, and quantifying the control effects of its key parameters is a crucial step in translating this technology from mechanistic research to engineering application. Based on the compressible Navier-Stokes equations and using a helicopter rotor system as the computational case, numerical simulations of dynamic stall on the retreating rotor blade were conducted using a spanwise single-row blowing method on the upper surface of the blade. Through two-dimensional flow-field analysis, with the balance between lift and lift-to-drag ratio as the optimization objective, an optimal combination of parameters for the chordwise position and width of the blowing holes and the blowing momentum coefficient was determined. The results show that when the blowing velocity exceeds the freestream velocity, the blowing holes are located near the flow separation point and within a certain chordwise range downstream of it, and the width is moderate, the aerodynamic performance of the rotor system can be significantly improved. Furthermore, with lift maximization as the objective, the spanwise number and distribution pattern of the blowing holes were optimized based on three-dimensional flow-field analysis. It was found that the control scheme with a single row of uniformly distributed blowing holes arranged in the spanwise interval of 0.55R-0.75R delivers the best effect, achieving a maximum lift increase of 33.7% in the computational case of this study.

Full Text

Research on the Radial Blowing Method for Dynamic Stall Control of Retreating Rotor Blades

LIU Muqi^1, WU Fengfeng^1,2, HALAHAN M^2, LIAO Wenyi^1, LI Yi^1

(1. School of Aeronautics and Astronautics, Taiyuan University of Technology, 030600 Jinzhong; 2. Technical Research and Development Department, Shanxi Zhehang Aviation Industry Co., Ltd., 030008 Taiyuan)

Abstract: Blowing control is one of the effective technical approaches for suppressing dynamic stall on helicopter rotor retreating blades. Quantifying the control effects of its key parameters is a critical step in transforming this technology from mechanistic research to engineering application. Based on the compressible Navier–Stokes equations, this study takes a certain helicopter rotor system as the computational case and adopts a single spanwise row of blowing orifices on the upper surface of the blade to numerically investigate dynamic stall on the rotor retreating blade. Through two-dimensional flow-field analysis, with the optimization objective of balancing lift and lift-to-drag ratio, an optimal combination of the chordwise location, width, and blowing momentum coefficient of the blowing orifices is determined. The results show that when the blowing velocity exceeds the freestream velocity, the blowing orifices are located near the flow-separation point and within a certain chordwise range downstream of it, and the width is moderate, the aerodynamic performance of the rotor system can be significantly improved. Further, with lift maximization as the objective, three-dimensional flow-field analysis is used to optimize the spanwise number and distribution pattern of the blowing orifices. It is found that the control scheme with a single uniformly distributed row of blowing orifices arranged over the blade spanwise interval of $0.55R \sim 0.75R$ is optimal; in the present computational case, a maximum lift increase of 33.7% can be achieved.

Keywords: rotor; blowing control; dynamic stall; retreating blade; aerodynamic performance

Chinese Library Classification No.: V211.3 **Document Code:** A **DOI:** 10.11776/j.issn.1000-4939.2026.03.007

Article No.: 1000-4939(2026)03-0571-11 **CSTR:** 32414.14.cjam.issn.1000-4939.2026.03.007

Research on the Radial Blowing Method for Dynamic Stall Control of Backward Propeller Blades

LIU Muqi^1, WU Fengfeng^1,2, HALAHAN M^2, LIAO Wenyi^1, LI Yi^1

(1. School of Aeronautics and Astronautics, Taiyuan University of Technology, 030600 Jinzhong, China;
2. Technical Research and Development Department, Shanxi Zhe Hang Aviation Industry Co., Ltd., 030008 Taiyuan, China)

Abstract: Blowing control is an effective technique to suppress dynamic stall on helicopter rotor retreating blades. Quantifying the control effects of key parameters is a critical step in translating this technology from mechanistic research to engineering application. This study numerically investigates its application using the compressible Navier–Stokes equation. A spanwise blowing slot on the blade' s upper surface is adopted. Two-dimensional analysis first optimizes blowing location, width, and momentum to balance lift and lift-to-drag ratio. Results show that blowing velocities exceeding the freestream, with the slot properly located near or within a certain chordwise range aft the separation point, significantly improve aerodynamic performance. Further three-dimensional analysis maximizes lift by optimizing the spanwise distribution of blowing

---

Date received: 2025-11-28

Fund project: Project supported by the Shanxi Provincial Major Science and Technology Special Plan (No. 202301120401010)
Corresponding author: WU Fengfeng, associate professor. E-mail: wffqll00@163.com
Citation format: LIU Muqi, WU Fengfeng, HALAHAN M, et al. Research on the radial blowing method for dynamic stall control of retreating rotor blades[J]. *Chinese Journal of Applied Mechanics*, 2026, 43(3): 571–581.
LIU Muqi, WU Fengfeng, HALAHAN M, et al. Research on the radial blowing method for dynamic stall control of backward propeller blades[J]. *Chinese Journal of Applied Mechanics*, 2026, 43(3): 571–581.

holes. A single uniform row within $0.55R \sim 0.75R$ is identified as optimal, achieving up to a 33.7% lift increase in the present case.

Key words: rotor; blowing control; dynamic stall; retreating blade; aerodynamic performance

During forward flight of a helicopter, the rotor is subjected to asymmetric incoming flow. To maintain moment balance, the retreating blade must increase its pitch to compensate for the reduction in incoming-flow velocity. However, during high-speed forward flight, the retreating blade is in a state of relatively low speed and large angle of attack; significant flow separation occurs on the upper surface of the airfoil, inducing a dynamic stall vortex (DSV). As the rotor rotates, the blade angle of attack and the incoming-flow velocity vary periodically. The DSV undergoes an unsteady process of generation, shedding, and reattachment, thereby causing a class of dynamic-stall problems[1]. This problem not only leads to severe aerodynamic oscillations and abrupt lift changes, aggravating rotor vibration and structural fatigue, but also, because the sharp reduction in lift on the retreating blade destroys the rolling-moment balance, induces control-reversal effects and thus severely constrains the forward-flight speed and maneuverability of helicopters[2]. Therefore, developing effective flow-control methods to suppress dynamic stall is of important research value for improving the flight performance and safety of helicopters[3].

To achieve effective suppression of dynamic stall, existing flow-control methods are mainly divided into passive control and active control. Passive control techniques are mainly realized by optimizing the blade airfoil geometry or installing fixed mechanical structures on its surface[4–5]. By altering the geometry of the airfoil leading edge[6–7] and trailing edge[8], the delay or advance of dynamic stall caused by changes in resistance can be effectively regulated. The study by Shen Simeng et al.[9] showed that geometric parameters such as airfoil thickness and camber, by regulating the evolution and intensity of the dynamic stall vortex and the trailing-edge vortex, have a significant influence on the secondary aerodynamic-force peak, and that changes in geometric parameters systematically affect its amplitude and phase. In addition, installing vortex generators[10], spoilers[11], or cavities with specific geometric parameters[12] on the upper surface of the blade can suppress flow separation and delay stall onset, thereby achieving effective control of dynamic stall. Although such passive control methods have the advantage of high reliability, their control effects are significantly constrained by the device size and installation position. Larger devices not only introduce additional drag but also increase structural complexity; moreover, because they are essentially static control measures, their performance in coping with highly unsteady flows is often unsatisfactory[13].

In contrast, active flow control relies on external energy input. Typical methods include plasma actuation, active control with small trailing-edge flaps[13], combined blowing-suction control[14], blowing[15], and synthetic jets[16–17], all of which possess the distinctive advantage of responding dynamically to flow variations. Among them, blowing control suppresses the DSV by injecting high-energy fluid, and has unique advantages such as high control efficiency and strong environmental adaptability. The Fairey Rotodyne demonstrator of the British company Fairey is an engineering example of its successful application[18]. GARDNER et al.[19–20] further verified its effectiveness through stall experiments. Their research showed that using a narrow-slot jet near the leading edge of the airfoil can increase the maximum stall angle of attack by about 4° and significantly reduce the peak pitching moment by as much as 65%. However, the effectiveness of blowing control depends strongly on the reasonable setting of key parameters. MÜLLER-VAHL et al.[21] emphasized that the blowing velocity must be higher than the maximum boundary-layer velocity; otherwise, leading-edge separation may be intensified. In addition, the blowing momentum coefficient, blowing-hole location, and their chordwise and spanwise distributions all have decisive influences on the control effect[22]. Notably, SINGH et al.[23] investigated a control method based on spanwise oblique blowing arrays, and found that a blowing array arranged at the leading edge can effectively suppress the generation of the dynamic stall vortex. However, their study failed to provide key design parameters such as the optimal arrangement of the array, which restricts the engineering application of this technology; related optimization issues remain to be solved.

This study takes a certain type of helicopter rotor as the object and, based on the computational fluid dynamics (CFD) method, systematically investigates the suppression effect of a spanwise single-row blowing jet arranged on the upper surface of the blade on dynamic stall, as well as its influence on aerodynamic performance. First, two-dimensional flow-field analysis is carried out; taking the momentum coefficient as the core variable, key parameters such as the chordwise position and width of the blowing hole and their matching relationships are optimized and determined. A three-dimensional rotor model is then constructed, and spanwise arrangements of blowing holes are designed and compared. The aim of this study is to reveal the mechanism by which spanwise blowing control acts on dynamic stall of the retreating blade, as well as its potential for aerodynamic benefit, so as to provide a reference for early-stage novel aerodynamic design of rotor systems.

1 Numerical Simulation Method and Model

1.1 Governing equations for the unsteady flow field of the rotor

To accurately simulate the unsteady, compressible flow field around the helicopter rotor and capture the evolution of its complex vortex structures, a flow-control model is established in an inertial coordinate system. The flow process is governed by the compressible unsteady Navier-Stokes (N-S) equations[24], namely

$$\frac{\partial}{\partial t}\int_{\Omega}\mathbf{W}\mathrm{d}\Omega+\oint_{\partial\Omega}\left(\mathbf{F}_c^M-\mathbf{F}_v\right)\cdot\mathbf{n}\mathrm{d}S=0 \tag{1}$$

where $\mathbf{W}=[\rho,\rho u,\rho v,\rho\varpi,\rho e]^{\mathrm{T}}$ is the vector of conserved variables; $\mathbf{F}_c^M=(f,g,h)$ and $\mathbf{F}_v=(a,b,c)$ are the convective flux and viscous flux, respectively; $\mathbf{n}$ is the unit normal vector; and Ω is the control-volume volume. The specific expressions for the relevant parameters are as follows.

$$f=\begin{bmatrix}\rho(u-u_\omega)\\ \rho u(u-u_\omega)+p\\ \rho v(u-u_\omega)\\ \rho\varpi(u-u_\omega)\\ \rho h(u-u_\omega)+pu_\omega\end{bmatrix},\qquad g=\begin{bmatrix}\rho(v-v_\omega)\\ \rho u(v-v_\omega)\\ \rho v(v-v_\omega)+p\\ \rho\varpi(v-v_\omega)\\ \rho h(v-v_\omega)+pv_\omega\end{bmatrix},$$

$$h=\begin{bmatrix}\rho(\varpi-\varpi_\omega)\\ \rho u(\varpi-\varpi_\omega)\\ \rho v(\varpi-\varpi_\omega)\\ \rho\varpi(\varpi-\varpi_\omega)+p\\ \rho h(\varpi-\varpi_\omega)+p\varpi_\omega\end{bmatrix},$$

$$a=\begin{bmatrix}0\\ \tau_{xx}\\ \tau_{yx}\\ \tau_{zx}\\ u\tau_{xx}+v\tau_{yx}+\varpi\tau_{zx}-q_x\end{bmatrix},\qquad b=\begin{bmatrix}0\\ \tau_{xy}\\ \tau_{yy}\\ \tau_{zy}\\ u\tau_{xy}+v\tau_{yy}+\varpi\tau_{zy}-q_y\end{bmatrix}, \tag{2}$$

$$c=\begin{bmatrix}0\\ \tau_{xz}\\ \tau_{yz}\\ \tau_{zz}\\ u\tau_{xz}+v\tau_{yz}+\varpi\tau_{zz}-q_z\end{bmatrix}.$$

where ρ is the gas density; u, v, and ϖ are, respectively, the three components of the relative velocity vector of the rotor-grid cell in the inertial coordinate system; u_ω, v_ω, and ϖ_ω are, respectively, the three velocity components of the rotor grid in the three directions, i.e., the entrainment velocity; p is the pressure; e is the total energy per unit mass; and h is the total enthalpy.

In the viscous flux, the stress expressions are

$$\begin{aligned}
\tau_{xx} &= \frac{2}{3}\mu\left(2\frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} - \frac{\partial \varpi}{\partial z}\right), \quad \tau_{xy} = \tau_{yx} = \mu\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right), \quad q_x = -k\frac{\partial T}{\partial x}, \\
\tau_{yy} &= \frac{2}{3}\mu\left(2\frac{\partial v}{\partial y} - \frac{\partial u}{\partial x} - \frac{\partial \varpi}{\partial z}\right), \quad \tau_{yz} = \tau_{zy} = \mu\left(\frac{\partial \varpi}{\partial y} + \frac{\partial v}{\partial z}\right), \quad q_y = -k\frac{\partial T}{\partial y}, \\
\tau_{zz} &= \frac{2}{3}\mu\left(2\frac{\partial \varpi}{\partial z} - \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y}\right), \quad \tau_{xz} = \tau_{zx} = \mu\left(\frac{\partial \varpi}{\partial x} + \frac{\partial u}{\partial z}\right), \quad q_z = -k\frac{\partial T}{\partial z}.
\end{aligned} \tag{3}$$

where μ is the viscosity coefficient, k is the thermal conductivity coefficient, and T is the temperature.

For time advancement, the implicit LU-SGS method is adopted, namely the LU-SSOR method. The turbulence model selected is the k-ω SST model, which has relatively good accuracy and robustness in predicting flow separation with adverse pressure gradients, and has been widely used in engineering studies on dynamic stall of airfoils [25].

For ease of analysis and calculation [26], the momentum coefficient C_μ is defined as

$$C_\mu = 2\frac{h}{c}\left(\frac{U_\mathrm{m}}{\sqrt{2U_\infty}}\right) \tag{4}$$

where U_m is the blowing-gas velocity, U_∞ is the incoming-flow velocity, h is the blowing-slot outlet width, and c is the airfoil chord length.

1.2 Establishment and Validation of the Simulation Model

This study takes the NACA 23012 rotor-blade airfoil of a certain vertical-takeoff aircraft as the research object. A chord length of 150 mm is selected, and numerical simulations are carried out under the flow condition of Reynolds number $Re = 1.2 \times 10^6$. The computational domain is set as follows: the left boundary and the upper and lower boundaries are each 20 chord lengths from the airfoil, and the right boundary is 30 chord lengths behind the trailing edge of the airfoil. To accurately capture the vortex structures in the flow around the airfoil, a gradually varying mesh design from dense to sparse is adopted: prism-layer grids are arranged in the near-wall region and refined; the grid spacing within the wall boundary layer is set to 10^{-4} times the chord length, with $y^+ < 2$, satisfying the requirements of the RANS method; the grid growth rate is 1.05. In the region away from the wall, a gradually coarsened polygonal mesh is adopted. According to the hole-mesh refinement method proposed in Ref. [27], local mesh refinement is performed for the blowing hole and its surrounding region, ensuring that the mesh size transitions smoothly along the blowing direction [27 − 29]. Figure 1 compares the variation of lift coefficient with angle of attack under prism-layer grids of 10 layers (Grid 1), 20 layers (Grid 2), and 30 layers (Grid 3). Considering both accuracy and efficiency, the 20-layer mesh ensures accurate prediction of aerodynamic coefficients while also providing faster convergence and good numerical stability; therefore, a mesh node number of 225×100 is selected (Fig. 2).

(a) Grid 1 (b) Grid 2 (c) Grid 3

Fig. 1 Computational model schematic diagram

According to the comparison in Fig. 3, the difference between the trend of the experimental values combined with the wind-tunnel numerical simulation in Ref. [30] and the simulation results of this study is within 2%, indicating that this model and numerical simulation method have sufficient engineering accuracy within the parameter range of this study. Therefore, they are selected for all subsequent two-dimensional and three-

the baseline configuration for the three-dimensional simulation.

Fig. 2 Air blow-hole grid schematic diagram

Fig. 3 Comparison of reference values and simulated values
Plot annotations: Mesh 1; Mesh 2; Mesh 3; Reference [30]; vertical axis C_L; horizontal axis $\alpha/(^\circ)$.

2 Case Analysis

Three-dimensional numerical simulations were carried out for a four-bladed rotor system. The rotor diameter was 6 720 mm, the rotational speed was 500 rad/min, the blades were rectangular and untwisted, and the airfoil was the one used above. The rotor solidity was 0.066, and the incoming-flow velocity was 61.1 m/s; the rotor-disk tilt angle was 12°, and the collective pitch angle was 13°.

2.1 Two-Dimensional Flow-Field Analysis

Numerical simulations were conducted for the characteristic blade section, with the free-stream Mach number set to 0.44. In the Star-CCM+ software [31], the flow field around the airfoil was numerically simulated to obtain the evolution of the velocity vectors in the flow field at different instants, as shown in Fig. 4. Under unsteady dynamic conditions, that is, when the angle of attack varies periodically as $\alpha(t) = 7.5^\circ + 12.5^\circ \sin(\omega t)$ [32], where the angular frequency is

$\omega = 75.4$ rad/s and the reduced frequency is $k \approx 0.0377$. When the effective angle of attack of the airfoil exceeds the static stall angle of attack, the high adverse pressure gradient formed near the leading edge causes flow separation. A free shear layer forms downstream of the leading edge; this layer rapidly rolls up and forms vortical disturbances, leading to the gradual generation and development of a vortex structure (DSV). Under the influence of the upstream incoming flow, as this vortex structure sweeps rearward along the airfoil surface toward the trailing edge, its strong induced effect disrupts the flow equilibrium at the trailing edge, promoting the generation and shedding of a trailing-edge vortex (TEV) rotating in the opposite direction. The interaction between the two causes the vortex-system scale to increase and the flow-separation region to expand rapidly, and the airfoil consequently enters deep stall.

Fig. 4 Velocity vector diagram
Panel annotations: (a) $t = 0.2T\uparrow$; (b) $t = 0.4T\uparrow$; (c) $t = 0.6T\downarrow$.
$\uparrow$—pitch-up motion; $\downarrow$—pitch-down motion.
In-figure labels: DSV; TEV.

Based on the inviscid-flow assumption of blade-element theory, blowing control causes high-speed airflow to adhere to the airfoil surface through the Coandă effect and uses momentum injection to enhance the boundary layer' s ability to resist an adverse pressure gradient, thereby effectively suppressing flow separation. In an inviscid model, this process can be equivalently regarded as active control of the streamlines around the airfoil and of the circulation distribution, thereby optimizing the lift characteristics and surface pressure distribution. The blowing parameters have a significant influence on the control effect: when the blowing-slot width and velocity are too small, the injected momentum is insufficient to alter the boundary-layer structure, making it difficult to suppress flow separation; when the parameters are too large, however, the main flow is excessively disturbed, which may induce overly high local flow velocities and form shock waves on the airfoil surface, leading to a sharp increase in drag and deterioration of control performance [13]. To study the chordwise influence of the blowing-slot position relative to the separation point, this study combines Fig. 4 with the experiments in Ref. [25] and selects three typical positions for comparative analysis: 13%c from the leading edge (upstream of the separation point), 26%c (near the separation point), and 50%c (downstream of the separation point). According to the two-dimensional calculation results, the instant at which the vortex-structure scale is largest is selected for the analysis of the relevant aerodynamic data.

2.2 Evaluation of Parameter-Coupling Effects Based on Orthogonal Experiments and the Response-Surface Method

To carry out a systematic comparative analysis and ensure that the conclusions conform to the physical laws of engineering applications, this study selects the blowing momentum coefficient C_μ as the core nondimensional parameter characterizing blowing intensity, in order to quantitatively examine the coupling relationship among key parameters such as blowing velocity and blowing-slot width, and to verify the feasibility of the stepwise optimization strategy. Under the condition of a fixed chordwise position of the blowing slot near the airflow separation point, an orthogonal experimental design method is used to configure parameter combinations, and a response-surface analysis method is combined for parameter optimization, so as to evaluate the influence of coupling among parameters on the final optimization results [30]. Fig. 5 presents the aerodynamic-performance responses under different parameter combinations.

The results show that, near a blowing-slot width of $h = 2.5\%c$ and a momentum coefficient of $C_\mu = 1.3\%$, there exists a parameter region with relatively good aerodynamic performance. The favorable parameter combinations determined in the subsequent study are all distributed within this range, preliminarily verifying the rationality of the coupling analysis and the stepwise optimization pathway.

Fig. 5 Aerodynamic performance corresponding to each parameter group / Aerodynamic performance for different parameter groups
(a) L/D; (b) L/N

2.3 Blowing-slot width and velocity

The coupling relationship between blowing velocity and blowing-slot width is considered comprehensively. With the momentum coefficient fixed at $C_\mu = 1.3\%$, the blowing-slot width is adjusted to coordinate the matching relationship between the two; the results are shown in Fig. 6. Compared with the baseline configuration without slots ($L/D = 6,\ L = 3\,161$ N), when the blowing-slot width is in the range of $0.2\%c$-$1\%c$, the lift-to-drag ratio is relatively high, but the increase in lift is limited. When the width increases to the range of $1.2\%c$-$2.4\%c$, the lift increases significantly, but the lift-to-drag ratio decreases to some extent. When the width is around $2.5\%c$, the lift-to-drag ratio tends to be stable and the lift remains at a relatively favorable level. When the width exceeds $2.6\%c$, both lift and lift-to-drag ratio decrease markedly. Considering both lift performance and aerodynamic efficiency, a blowing-slot width of $2.5\%c$ is selected as the optimal configuration.

Fig. 6 Lift-to-drag ratio and lift of the airfoil as a function of blowing-slot width (h) / Lift-to-drag ratio and lift as a function of blowing slot width (h)
(a) h-L/D; (b) h-L

With the blowing-slot width fixed at $2.5\%c$, the influence of the blowing momentum coefficient on the airfoil lift-to-drag ratio (L/D) and lift (L) is further investigated; the results are shown in Fig. 7. The results indicate that when the blowing momentum coefficient is $C_\mu = 1.3\%$—corresponding to a blowing velocity of approximately 142% of the freestream velocity—the flow-control effect is optimal, and the airfoil exhibits the best overall aerodynamic performance.

Fig. 7 Lift-to-drag ratio and lift of the airfoil for different momentum coeffi-

cients / Lift-to-drag ratio and lift as a function of momentum
(a) C_μ-L/D; (b) C_μ-L

2.4 Chordwise distribution of blowing slots

Figure 8 presents the influence of three blowing-slot locations on the control of the airfoil aerodynamic characteristics. It can be seen from the figure that the blowing excitation at 26%c and 50%c provides better control than the blowing excitation at 13%c. After control is applied, the lift coefficient increases, the drag coefficient decreases, and the area of the hysteresis loop is reduced. This indicates that blowing injection effectively increases the circulation around the airfoil and effectively controls unsteady aerodynamic losses.

Fig. 8 Influence of different blowing locations on the control effect of airfoil aerodynamic characteristics / Comparison of baseline and dynamic test data
(a) Lift-coefficient curves; (b) Drag-coefficient curves

Figure 9 shows the velocity-streamline contours of the time-averaged flow field around the airfoil for $C_\mu = 1.3\%$ and $h = 2.5\%c$, with the blowing-velocity direction parallel to the freestream direction and the blowing slots located at 13%c, 26%c, and 50%c, respectively.

Unit: $\mathrm{m \cdot s^{-1}}$

Fig. 9 Velocity-streamline contours for different blowing locations
(a) $C_\mu = 1.3\%,\ 13\%c$; (b) $C_\mu = 1.3\%,\ 26\%c$; (c) $C_\mu = 1.3\%,\ 50\%c$

When the blowing hole is arranged at 13%c, the high-speed blowing causes the separation point to move slightly downstream, but the downstream shift is limited and cannot effectively suppress the subsequent flow separation. The area of the separation region does not decrease noticeably, and the strength of the trailing-edge stall vortex is not effectively controlled (Fig. 9a). Meanwhile, new separation vortices continue to form in the leading-edge region. These vortex structures develop downstream with the main flow and continuously intensify, ultimately shedding from the trailing edge. The aerodynamic data shown in Fig. 10 further indicate that this blowing location has no significant effect on improving the overall aerodynamic performance of the airfoil.

Fig. 10 Velocity streamline contours at the 50%c blowing location
Unit: $\mathrm{m \cdot s^{-1}}$; color scale: 0–400.
(a) $C_\mu = 1.4\%$; (b) $C_\mu = 1.5\%$.

When the blowing hole is arranged at 26%c, the flow-field control effect is significant. The streamline results show (Fig. 9b) that blowing has little influence on the leading-edge region, where the airflow displacement thickness is basically consistent with that in the uncontrolled state. In the middle and rear portions of the airfoil, however, the displacement thickness on the upper surface is significantly reduced, indicating weakened vortical motion within the boundary layer and increased flow velocity. This enhances the ability of the boundary layer to resist the adverse pressure gradient, effectively delays flow separation, and local flow reattachment contributes to improved aerodynamic performance of the airfoil. The separation point shifts downstream of the blowing hole, and the formation of the flow-separation vortex is correspondingly delayed. The laminar-flow region on the upper surface of the airfoil expands, ultimately increasing the airfoil lift by about 51% and improving the lift-to-drag ratio by about 31%.

When the blowing hole is arranged at 50%c, the injection of high-energy fluid does not cause the separation point to move downstream (Fig. 9c). However, the numerical simulation results show that implementing blowing control at this location and increasing the blowing velocity can significantly improve the aerodynamic performance of the airfoil: the lift coefficient increases by more than 34%, and the lift-to-drag ratio increases by more than 24%, as shown in Fig. 11. The analysis suggests that, because the separation point is already upstream of the blowing hole, blowing cannot suppress the onset of separation, but it can effectively intervene in the development and evolution of the already formed vortex structures by enhancing the energy input in the near-wall region, thereby improving flow reattachment over the middle and rear portions.

Fig. 11 Lift-to-drag ratio and lift as a function of blowing-hole location
(a) L/D; (b) L. Blowing-hole locations: no hole, 13%c, 26%c, 50%c.

From the comparison in Fig. 12, it can be seen that the greater the blowing velocity, the smaller the area of the hysteresis loop, the smoother the curve transition, the larger the lift coefficient, and the smaller the drag coefficient. This indicates that the improvement becomes increasingly significant as the blowing velocity increases, and the control effect on dynamic stall becomes better.

Fig. 12 Comparison of baseline and dynamic test data
(a) Lift-coefficient curve; (b) Drag-coefficient curve.
Legend: baseline; 50%c, $C_\mu = 1.4\%$; 50%c, $C_\mu = 1.5\%$.

When the blowing-hole location is moved downstream to the critical position of 74%c, the blowing-control effect is significantly weakened. The numerical simulation results show (Fig. 13) that, under this arrangement, blowing cannot effectively reduce the displacement thickness on the airfoil surface, nor can it effectively suppress the evolution of the trailing-edge vortex. Because the blowing-injection position is too far aft, it can hardly exert effective control over the flow disturbance, and the aerodynamic performance cannot be improved.

Fig. 13 Velocity streamline contours of the airfoil at 74%c
Unit: $\mathrm{m \cdot s^{-1}}$; color scale: 0–400.

Based on comprehensive consideration of control effectiveness and engineering feasibility, the chordwise location of the blowing hole is recommended to be arranged from near the separation point to the middle and rear regions of the airfoil. As verified in Sections 2.2–2.4, the conclusions of this section are also applicable from the airflow separation point to the middle and rear portions of the airfoil. In subsequent research, two locations, 26%c near the separation point and 50%c downstream of the separation point, are selected for three-dimensional numerical simulation, in order to further investigate the actual control effect of the blowing-control strategy in a three-dimensional rotor environment.

3 Three-Dimensional Flow-Field Simulation

To simplify the calculation, the blade is modeled as a flapless configuration, and periodic pitch control is introduced accordingly [26]. Because the two-dimensional model neglects real physical effects in three-dimensional rotor flow, such as centrifugal force, Coriolis force, and spanwise pressure gradients, systematic research is further carried out in a three-dimensional rotor model on the basis of the two-dimensional parameter optimization: first, the retreating-blade span···

the typical spanwise separation interval ($0.55R \sim 0.75R$). The blowing-hole locations selected from the two-dimensional optimization (26%c, 50%c) and the momentum coefficient ($C_\mu = 1.3\%$) were then taken as the baseline configuration, with emphasis on examining the influence of the spanwise arrangement—long single slot versus single-row distributed blowing—on aerodynamic performance.

3.1 Stall-State Calculation

Based on rotor kinematic analysis, the local reduced frequency at the characteristic section was determined to be $k \approx 0.044$ [24]. Figure 14 shows the corresponding distribution of the rotor flow velocity.

Fig. 14 Flow velocity distribution on the retreating side of the rotor ($0.55R \sim 0.75R$)

Unit: $\mathrm{m \cdot s^{-1}}$; color scale: -50 to 200. Azimuth labels: $\psi = 180°$, $\psi = 270°$, $\psi = 90°$, $\psi = 0°$.

Figure 15 shows the velocity-streamline distributions of the characteristic blade section at azimuth angles from 180° to 360°. Because of the presence of unsteady effects, flow separation on the upper surface of the retreating-side blade is delayed. Together with the weakening of the adverse pressure gradient, this produces delayed boundary-layer separation and thereby increases the effective angle of attack, as shown in Fig. 15(a). Under dynamic conditions, the strong adverse pressure gradient formed near 13%c at the leading edge induces flow separation. The free shear layer formed downstream of the leading edge rolls up rapidly and evolves into a concentrated vortex structure. The velocity distribution on the retreating side is markedly nonuniform; relatively strong vortex structures appear within $225° \sim 270°$, and are especially severe near the 270° azimuth angle, indicating that this region exhibits pronounced dynamic-stall characteristics.

Figure 16 shows the vorticity distribution on the upper surface over the characteristic-section interval $0.55R \sim 0.75R$ during the motion of the retreating blade. In this case, the combined effects of upstream wake shedding and the unsteady boundary-layer response in this region induce flow separation and promote the formation of vortex disturbances from the leading-edge region near 13%c to downstream areas, sweeping toward the trailing edge and blade tip. After the vortex structures sweep downstream and merge into the turbulent wake field, the airflow over the blade upper surface enters a state of complete separation. This series of flow evolutions causes the effective angle of attack at each section to vary significantly with azimuth angle, leading to a nonuniform distribution of rotor-disk loading. In particular, at the 270° azimuth angle, the blade is under a "large-angle-of-attack" condition, inducing the formation and shedding of leading-edge vortices and thus causing typical dynamic-stall phenomena such as abrupt lift changes; therefore, blowing control is applied in this interval.

Unit: $\mathrm{m \cdot s^{-1}}$; color scale: 0 to 400.

Subpanels: (a) $\Psi = 180°$; (b) $\Psi = 225°$; (c) $\Psi = 235°$; (d) $\Psi = 245°$; (e) $\Psi = 250°$; (f) $\Psi = 255°$; (g) $\Psi = 260°$; (h) $\Psi = 265°$; (i) $\Psi = 270°$; (j) $\Psi = 360°$.

Fig. 15 Velocity streamline contours for various blade sections at different azimuth angles ($0.7R$)

Subpanels:
(a) Vorticity distribution before the retreating blade passes azimuth angle 270°
(b) Vorticity distribution after the retreating blade passes azimuth angle 270°

Unit: $\mathrm{s^{-1}}$; color scale: 0 to 1×10^6.

Fig. 16 Vorticity contour at the $0.55R \sim 0.75R$ blade section on the rotor retreating side

3.2 Spanwise blowing-hole configuration

3.2.1 Long single-slot blowing

In the interval $0.55R \sim 0.75R$, where the blade characteristic sections are located, blowing holes with a width of 2.5%c and a length of $0.2R$ are arranged (Fig. 17).

(a) 13%c (b) 26%c (c) 50%c

Fig. 17 Blowing-hole locations
Fig. 17 Blowing location for airfoil

According to the preceding analysis results, aerodynamic data were analyzed when the vortex-structure scale is largest. In the three-dimensional rotor flow field, the control effects of three blowing-hole locations–13%c, 26%c, and 50%c –were compared. The results are shown in Figs. 18 and 19 and Table 1. Among them, two arrangements can suppress the vortex disturbance in a concentrated manner, delay the formation of the vortex structure, weaken the dynamic interference of the advancing-blade wake on the retreating blade, and improve the overall aerodynamic performance. Specifically, when the blowing holes are located at 26%c and 50%c, the aerodynamic gain is further increased, and the lift and lift-to-drag ratio can be increased by about 15% and 13%, respectively.

(a) 13%c

(b) 26%c

(c) 50%c

Unit: s^{-1}

Fig. 18 Vorticity contours of the improved retreating-side rotor blade
Fig. 18 Vorticity contour of improved retreating rotor blade

Table 1 Rotor lift-to-drag ratio and lift under long single-slot blowing
Tab. 1 Rotor lift-to-drag ratio and lift as a function of long slot blowing

Parameter	Slot relative position: no slot	Slot relative position: 13%c	Slot relative position: 26%c	Slot relative position: 50%c
$\frac{L}{D}$	5	5	6	6
L/N	10 457.8	10 553.9	11 973.9	12 046.4

(a) 13%c (b) 26%c (c) 50%c

Labels shown in each rotor disk: $\psi = 180°$, $\psi = 270°$, $\psi = 90°$, and $\psi = 0°$.
Unit: $m \cdot s^{-1}$; color scale: -50 to 200.

Fig. 19 Rotor airflow velocity distribution after improvement
Fig. 19 Rotor airflow velocity distribution after improvement

3.2.2 Single-row distributed blowing

Within the blade spanwise range $0.55R \sim 0.75R$, several rectangular blowing holes with the same width of 2.5%c are uniformly arranged. The length of a single blowing hole is denoted by a, and its specific size is determined by $a = x \cdot A$, $A = 0.2R/n$. Here, n is the number of blowing holes, A is the unit length obtained by equally dividing the spanwise control interval, and x is the ratio of the blowing-hole length a to A, defined as the aspect ratio (Fig. 20).

Fig. 20 Schematic diagram of the blowing-hole aspect ratio
Fig. 20 Schematic diagram of air injection hole aspect ratio

The momentum coefficient is taken as 1.3%. Under the condition of a fixed momentum coefficient, blowing holes are arranged at the chordwise locations 26%c and 50%c, respectively, and 3, 4, and 5 blowing holes are configured at each location for comparative study. The calculation results show that, under all operating conditions, the lift-to-drag ratio lies in the range 5.49 ∼ 6.91, which belongs to the typical aerodynamic-efficiency range. To further analyze the influence of blowing-hole number and location on the control effect, the corresponding lift-performance data are summarized in Tables 2 and 3.

From the comparative analysis of Tables 2 and 3, it can be seen that, in terms of aerodynamic-performance data, compared with the long single-slot blowing scheme, the single-row distributed blowing-hole arrangement can more significantly improve the aerodynamic characteristics of the rotor in the stalled state. At the same time, the control results for different blowing-hole distributions also indicate that a larger number of excitation sources does not necessarily yield a better control effect on the aerodynamic characteristics of the rotor.

the better. The key lies in accurately identifying the location of the airflow separation point on the blade surface, and in reasonably setting the number of blowing holes and the hole width near it, so as to control rotor dynamic stall and airflow separation in a more targeted manner (Fig. 21).

Table 2 Rotor lift at 26%c for different numbers of blowing holes and different aspect ratios
Tab. 2 Rotor lift at 26%c as a function of hole number and aspect ratio

x	L/N, $n=3$	L/N, $n=4$	L/N, $n=5$
0.1	11 289.8	11 222.6	10 505.0
0.2	11 393.1	11 513.8	10 860.5
0.3	11 502.3	11 684.2	11 567.3
0.4	11 619.3	11 765.2	11 884.5
0.5	11 690.6	11 861.1	11 953.6
0.6	11 835.7	11 876.9	12 108.4
0.7	12 185.4	12 244.9	13 891.9
0.8	12 264.5	13 991.2	13 598.7
0.9	12 502.5	14 132.9	13 641.2

Table 3 Rotor lift at 50%c for different numbers of blowing holes and different aspect ratios
Tab. 3 Rotor lift at 50%c as a function of hole number and aspect ratio

x	L/N, $n=3$	L/N, $n=4$	L/N, $n=5$
0.1	11 154.4	11 079.7	11 250.0
0.2	11 016.2	11 273.9	11 362.5
0.3	12 255.7	11 386.0	11 453.5
0.4	12 326.1	11 682.6	11 700.9
0.5	12 680.7	11 660.7	13 666.7
0.6	12 724.6	11 954.2	13 790.0
0.7	12 082.2	12 269.1	13 986.6

x	L/N, $n = 3$	L/N, $n = 4$	L/N, $n = 5$
0.8	12 319.8	12 454.2	14 088.1
0.9	13 161.5	12 747.3	13 921.4

In the long single-slot blowing mode, the excitation region of the high-energy fluid acting on the mainstream is relatively concentrated. Although it shows a strong local flow-control effect, it can easily lead to nonuniform control, so that the airflow problems along the blade span cannot be effectively improved. By contrast, the single-row distributed blowing-hole scheme can achieve relatively uniform flow excitation and improve the overall aerodynamic performance. In addition, with consideration of structural strength, by reasonably setting the combination of distribution parameters for the blowing holes, the control effect of the blowing array on the rotor aerodynamic characteristics can be further enhanced while ensuring structural integrity.

(a) 26%c (b) 50%c

Fig. 21 Rotor lift comparison at different slenderness ratios (x)

4 Conclusion

Based on the compressible unsteady N-S equations, this study systematically investigated spanwise single-row blowing control for dynamic stall of the retreating blade of a helicopter rotor during preliminary aerodynamic design exploration. The following conclusions were obtained.

1) The two-dimensional flow-field analysis shows that the blowing-control effect depends critically on the proper matching of blowing parameters. When the blowing velocity is higher than the freestream velocity, the blowing holes are arranged near the airflow separation point and in its downstream region, and the hole width is moderate, the aerodynamic performance of the rotor airfoil during dynamic stall can be significantly improved. In the case studied here, when the blowing velocity is 142% of the freestream velocity, the blowing holes are located from 26%c to the middle–rear region, and the width is 2.5%c, the control effect is optimal.

2) The three-dimensional flow-field analysis further verifies that the parameter configuration optimized from the two-dimensional study still has a good control effect in a three-dimensional environment, as well as the effectiveness of two-dimensional analysis for screening key parameters. At the same time, the three-dimensional results further reveal the significant influence of the spanwise distribution mode on the control effect. A single-row uniformly distributed multi-hole blowing scheme, when the number of holes and the hole width are adapted according to the excitation interval, is superior in overall control effect to continuous long single-slot blowing. This scheme can achieve more uniform flow excitation and is beneficial

for maintaining structural strength. In the present case, a single row of uniformly distributed blowing holes was arranged in the $0.55R \sim 0.75R$ interval of the blade ($n = 4, 5,\ x = 0.6 \sim 0.8$), and the maximum lift increase reached 33.7%, demonstrating good potential for engineering application.

3) Combining the calculation results for a certain helicopter rotor case, this study further quantified the matching relationship of key parameters in two-dimensional and three-dimensional conditions, providing reference value for the promotion of blowing-control technology in the aerodynamic design of helicopter rotor systems. Further work may couple multiple factors such as structural-design reliability, flight-mission profiles (e.g., hover, forward flight, and maneuvering states), and control complexity, and systematically optimize the number distribution and geometric parameters of blowing holes, so as to maximize engineering improvement.

the effect of improving the aerodynamic performance of helicopter rotors.

References:

[1] LEISHMAN J G. Principles of helicopter aerodynamics [M]. 2nd ed. Cambridge: Cambridge University Press, 2006.

[2] MENG Xuanshi, JU Enbo, YU Yi. Research progress on dynamic stall and its experimental methods and measurement techniques [J]. Acta Aerodynamica Sinica, 2025, 43(7): 1-27 (in Chinese).

[3] YANG Hesen, ZHAO Guangyin, LIANG Hua, et al. Research progress on influence factors of airfoil dynamic stall and flow control [J]. Chinese Journal of Aeronautics, 2020, 41(8): 119-144 (in Chinese).

[4] BAGAI A. Aerodynamic design of the X2 technology demonstrator main rotor blade [C]//Proceedings of the 64th Annual Forum of the American Helicopter Society. Montreal, Canada: American Helicopter Society, 2008: 29-44.

[5] ASHTON N, BILLARD F, MOULINEC C, et al. The numerical simulation of the flow over a EC145 helicopter fuselage using HPC facilities [C]//Proceedings of the 23rd International Conference on Parallel Computational Fluid Dynamics. Barcelona, Spain, 2011.

[6] YU Boping, LI Gaohua, XIE Ling, et al. Dynamic stall optimization design of rotor airfoil based on surrogate model [J]. Journal of Zhejiang University (Engineering Science), 2020, 54(4): 833-842 (in Chinese).

[7] LI Congcong, SHI Yongjie, XU Guohua, et al. Research on the forward flight performance of rotor based on variable-droop leading edge [J]. Journal of Northwestern Polytechnical University, 2021, 39(3): 668-674 (in Chinese).

[8] LIND A H, JONES A R. Unsteady aerodynamics of reverse flow dynamic stall on an oscillating blade section [J]. Physics of Fluids, 2016, 28(7): 077102.

[9] JING Simeng, ZHAO Guoqing, ZHAO Qijun. Numerical research on secondary peak of aerodynamic forces of airfoil under dynamic stall [J]. Journal of Nanjing University of Aeronautics & Astronautics, 2022, 54(2): 191-202 (in Chinese).

[10] HEINE B, MULLENERS K, JOUBERT G, et al. Dynamic stall control by passive disturbance generators [J]. AIAA Journal, 2013, 51(9): 2086-2097.

[11] KAUFMANN K, GARDNER A D, RICHTER K. Numerical investigations of a back-flow flap for dynamic stall control [C]//Contributions to the 18th STAB/DGLR Symposium. Cham: Springer International Publishing, 2014: 255-262.

[12] ZHANG Xinyi, SUN Xiaojing. Study on dynamic stall characteristics of the airfoil with locally moving surface [J]. Journal of Chinese Society of Power Engineering, 2022, 42(9): 821-828 (in Chinese).

[13] LI Guoqiang, ZHAO Xinhai, YI Shihe, et al. Research progress on rotor reverse flow and dynamic stall flow control methods [J]. Journal of Experiments in Fluid Mechanics, 2023, 37(4): 29-47 (in Chinese).

[14] JIA Tianhao, GAO Chao, XU Zeyang, et al. Numerical simulation study on dynamic stall of airfoil controlled by combined blow-suction [J]. Advances in Aeronautical Science and Engineering, 2023, 14(4): 39-46 (in Chinese).

[15] MÜLLER-VAHL H F, NAYERI C N, PASCHEREIT C O, et al. Dynamic stall control via adaptive blowing [J]. Renewable Energy, 2016, 97: 47-64.

[16] FENG J J, ZHU G J, LIN Y, et al. Control of dynamic stall of an airfoil by using synthetic jet technology [J]. Arabian Journal for Science and Engineering, 2020, 45(11): 9835-9841.

[17] ZHAO Guoqing, ZHAO Qijun, GU Yunsong, et al. Experimental investigation of synthetic jet control on large flow separation of airfoil during stall [J]. Chinese Journal of Theoretical and Applied Mechanics, 2015, 47(2): 351-355 (in Chinese).

[18] GIBBINGS D. Fairey Rotodyne [M]. Stroud, Gloucestershire: The History Press, 2009.

[19] GARDNER A D, RICHTER K, MAI H, et al. Experimental investigation of air jets for the control of compressible dynamic stall [J]. Journal of the American Helicopter Society, 2013, 58(4): 1-14.

[20] GARDNER A D, RICHTER K, MAI H, et al. Experimental investigation of high-pressure pulsed blowing for dynamic stall control [J]. CEAS Aeronautical Journal, 2014, 5(2): 185-198.

[21] MÜLLER-VAHL H F, STRANGFELD C, NAYERI C N, et al. Control of thick airfoil, deep dynamic stall using steady blowing [J].

AIAA Journal, 2015, 53(2): 277-295.

[22] WEAVER D, MCALISTER K W, TSO J. Suppression of dynamic stall by steady and pulsed upper-surface blowing [C] // 16th AIAA Applied Aerodynamics Conference. Reston, VA, USA: AIAA, 1998: AIAA 1998-2413.

[23] SINGH C, PEAKE D J, KOKKALIS A, et al. Control of rotorcraft retreating blade stall using air-jet vortex generators [J]. Journal of Aircraft, 2006, 43(4): 1169-1176.

[24] ZHAO Qijun, XU Guohua, WANG Bo. *Helicopter Aerodynamics* [M]. Beijing: Science Press, 2024.

[25] XUE Yangliu, ZHAO Xielin, SUN Wenjie. Numerical study of airfoil dynamic stall based on the SDES method [J]. Chinese Journal of Applied Mechanics, 2024, 41(5): 999-1006.

XUE Yangliu, ZHAO Xielin, SUN Weijie. Numerical simulation of airfoil dynamic stall based on SDES method [J]. Chinese Journal of Applied Mechanics, 2024, 41(5): 999-1006 (in Chinese).

[26] ZHAO Guoqing. CFD simulation of unsteady dynamic stall of helicopter rotors and its active flow control [D]. Nanjing: Nanjing University of Aeronautics and Astronautics, 2015.

[27] ZHI H L, ZHU Z H, LU Y J, et al. Aerodynamic performance enhancement of co-flow jet airfoil with simple high-lift device [J]. Chinese Journal of Aeronautics, 2021, 34(9): 143-155.

[28] ZHANG Wei, YE Zhou, LI Chun, et al. Influence of parabolic slots on the aerodynamic performance of wind-turbine airfoils [J]. Chinese Journal of Applied Mechanics, 2024, 41(6): 1282-1287.

ZHANG Wei, YE Zhou, LI Chun, et al. Influence of parabolic slot on aerodynamic performance of wind turbine airfoil [J]. Chinese Journal of Applied Mechanics, 2024, 41(6): 1282-1287 (in Chinese).

[29] ZHA G C, PAXTON C. A novel airfoil circulation augment flow control method using co-flow jet [C] // 2nd AIAA Flow Control Conference. Reston, VA, USA: AIAA, 2004: AIAA 2004-2208.

[30] SMITH J A, JOHNSON L K. Large-eddy simulations of the NACA23012 airfoil with laser-scanned ice shapes [J]. Journal of Aircraft, 2023, 60(2): 450-465.

[31] RUAN Yuan, ZHANG Heng, LUO Ben, et al. Aerodynamic design of the wing of an amphibious aircraft with a hybrid wing-body configuration [J]. Advances in Aeronautical Science and Engineering, 2025, 16(5): 69-78.

RUAN Yuan, ZHANG Heng, LUO Ben, et al. Wing aerodynamic design of amphibious aircraft with hybrid-wing-body configuration [J]. Advances in Aeronautical Science and Engineering, 2025, 16(5): 69-78 (in Chinese).

[32] LI Guoqiang, SONG Kuihui, QIN Chen, et al. Experimental study on active control of airfoil dynamic stall based on a trailing-edge flap [J]. Chinese Journal of Aeronautics, 2024, 45(3): 105-120.

LI Guoqiang, SONG Kuihui, QIN Chen, et al. Test on active control of airfoil dynamic stall based on trailing edge flap [J]. Chinese Journal of Aeronautics, 2024, 45(3): 105-120 (in Chinese).

(Editor: Lü Yin)

Note: Figure translations are in progress. See original paper for figures.