ISSN: 2977-0041

Research Article

Journal of Material Sciences and Engineering Technology

Flow Control and Performance Enhancement of VTOL UAVS Using Vortex Generators and Blowing Techniques: A Numerical Approach

Tuğrul Oktay[1*] and Ebru Yıldız Aydemir[2]

[1]*Aeronautical Engineering, Faculty of Aeronautics and Astronautics, Erciyes University, Kayseri, Turkey*

[2]*Aerospace Engineering, Faculty of Aeronautics and Astronautics, Kocaeli University, Kocaeli, Turkey*

[*]Corresponding author

Tuğrul Oktay, Aeronautical Engineering, Faculty of Aeronautics and Astronautics, Erciyes University, Kayseri, Turkey.

Received: October 01, 2025; **Accepted:** October 07, 2025; **Published:** October 16, 2025

ABSTRACT

In this study, the aerodynamic performance of a vertical take-off and landing (VTOL) unmanned aerial vehicle (UAV) is investigated through the application of both active and passive flow control techniques. Passive control is achieved via vortex generators, while active control is implemented using blowing methods. A novel UAV design incorporating both strategies has been developed, and multiple configurations have been evaluated using Computational Fluid Dynamics (CFD) simulations. The main objective of this work is to enhance the overall flight performance of the UAV by delaying flow separation, controlling the boundary layer, and increasing aerodynamic efficiency. Various geometrical and operational parameters such as blowing magnitude, speed, angle, location and shape and size of the blowing area, as well as the shape, size and placement of the vortex generators were explored. In the first stage, each of these parameters was individually implemented and tested on the VTOL UAV design to assess its influence on aerodynamic performance. In addition, various combinations of these parameters were systematically investigated to evaluate their interactive effects and overall contribution to flow control efficiency. Preliminary results demonstrate that the integrated use of active and passive methods significantly improves aerodynamic performance, especially in transition regimes and low-speed operations. The proposed research provides important scientific insight contributions to the aerodynamic optimization of modern VTOL UAVs.

Keywords: Active Flow Control Method, Passive Flow Control Method, UAV, VTOL, Vortex Generator, Blowing Method, CFD, Performance Optimization, Separation Control

Introduction

VTOL (Vertical Take-Off and Landing) aerial vehicles have become a frequently utilized type of aircraft today due to their ability to operate effectively in environments with limited runway availability, such as ship decks, mountainous terrain, and urban areas. Moreover, VTOL platforms offer key operational advantages such as rapid take-off and landing, as well as the ability to hover, which enables accurate targeting and engagement of stationary objectives. VTOL systems are utilized for a wide range of mission profiles: in military operations for reconnaissance, surveillance, logistics and supply, precision strike, electronic warfare, search and rescue (SAR), and evacuation; in industrial applications for inspection, mapping, and measurement; and in the civilian sector for emergency medical services (EMS), air taxi operations, and cargo transport.

During the design phase of an aircraft, it is expected that the designed vehicle will exhibit high performance. Numerous methods have been investigated and developed to enhance aircraft performance up to the present day. Among these methods, flow control techniques—frequently employed to improve aerodynamic efficiency or to regulate flow characteristics—can be cited as notable examples. In the literature, flow control methods are predominantly categorized into two types: active and passive flow control techniques [1-3]. In some sources, however, a third category referred to as reactive flow control methods is also defined [4]. While passive flow control systems do not require any external energy input, active flow control systems necessitate an external energy supply. Reactive systems, on the other hand, operate based on data acquired by sensors.

Citation: Tuğrul Oktay, Ebru Yıldız Aydemir. Flow Control and Performance Enhancement of VTOL UAVS Using Vortex Generators and Blowing Techniques: A Numerical Approach. J Mat Sci Eng Technol. 2025. 3(4): 1-7. DOI: doi.org/10.61440/JMSET.2025.v3.77

Flow control systems are utilized for various purposes such as delaying flow separation, managing turbulence, increasing lift, reducing drag, enhancing controllability and controlling leading-edge vortices [5,6]. Among the widely used flow control techniques, passive methods include gurney flaps, cavities, mechanical vortex generators, zigzag (inverted V-shaped) patterns, and boundary layer trips; while active methods comprise dielectric barrier discharge (DBD) actuators, synthetic jets, the integration of oscillating or rotating components into the system, localized thermal actuation (surface heating and cooling), acoustic excitation, and steady/unsteady blowing and suction techniques. In this context, improving the aerodynamic performance of VTOL wings through flow control methods has become increasingly important. Within the scope of this study, the implementation of vortex generators and blowing-based flow control techniques is planned to enhance the aerodynamic performance of VTOL aircraft.

Vortex generators are aerodynamic structures that generate streamwise vortices. These vortices, formed in the direction of the flow, entrain the flow toward the boundary layer, thereby improving the velocity profile within it. Vortex generators enhance aerodynamic performance by suppressing undesirable flow separation [7]. Typically characterized by a low aspect ratio, vortex generators are placed on lifting surfaces upstream of the separation point to prevent flow detachment and effectively energize the boundary layer [4]. Vortex generators initiate secondary flow motion and restructure the flow field, thereby reducing the separated flow region and contributing to drag reduction. However, if an excessively large vortex generator is used, although the separated flow region may shrink, significant alterations in the flow structure can occur, leading to an increase in drag (as illustrated in Figure 1). Low-profile vortex generators ($h/\delta \leq 1$, where h is the height of the vortex generator and δ is the boundary layer thickness) are more effective in significantly reducing the separated flow region [8].

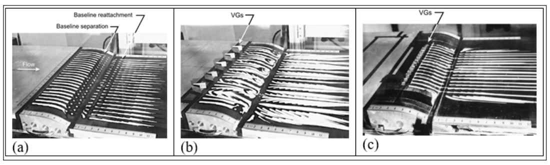

Figure 1: Oil flow visualizations showing the effect of the vortex generator on the flows (a) Without vortex generator, (b) Visualizations of wing-type counter-rotating vortex generators at a height of 0.8δ at 6 h above the base separation, (c) Visualizations of wing-type counter-rotating vortex generators at a height of 0.2δ at 10 h above the base separation [9].

In the study conducted by Kanat [2], the objective was to enhance performance through the design of a blowing system for unmanned aerial vehicles. In this context, compressed air was supplied via a compressor integrated into the fuselage of ZANKA II and ZANKA III, then directed through channels toward the blowing area located on the upper surface of the wing, from which it was discharged into the surrounding flow. Within the scope of the study, numerical simulations were conducted under identical environmental conditions for various configurations equipped with different blowing systems, in which the blowing region variables included the blowing area (shape and size), position relative to the leading edge, position relative to the wing root, blowing pressure magnitude, and blowing jet exit angle. The efficiency of the blowing systems was then evaluated based on the simulation results. The results indicate that, with the contribution of both the PID controller and the variable geometry mechanism, a 32% increase in the optimum aerodynamic efficiency was achieved, along with a 42% reduction in total energy consumption. Additionally, the implementation of the blowing system led to a reduction in shear stress.

Various studies have been conducted to investigate the effects of geometric and dynamic parameters of blowing systems on the aerodynamic performance of unmanned aerial vehicles (UAVs). In studies evaluating the influence of blowing region length, configurations with different spanwise extents were analyzed, and it was reported that the system applied along the entire wingspan provided the highest efficiency, resulting in a 240% increase in maximum aerodynamic efficiency [10]. In another study, the effect of blowing pressure magnitude was examined using six blowing regions positioned at 30%, 60%, and 90% of the half-span [11]. Although no linear relationship was observed, a 33% performance improvement was achieved at a blowing pressure of 25,000 Pa. In proceedings, analyzing the effects of blowing system parameters such as location, radius, and pressure, optimization approaches integrated with flight control systems were employed, aiming to minimize a defined cost function [12-13]. Simulation results showed that the most effective configuration featured a smaller radius ($r = 0.04$ m), placement closer to the trailing edge (0.7x/c), and a lower pressure (10,000 Pa), achieving up to a 60% increase in maximum aerodynamic efficiency, and improvements of 10% and 42% in lateral and longitudinal cost functions, respectively. In studies investigating the influence of blowing angle, various angles (0°, 45°, 90°) were tested [14]. It was emphasized that the blowing angle alone did not cause a significant difference and should be evaluated in conjunction with other parameters. Similarly, in another study, different angles, pressure levels (10,000 Pa, 25,000 Pa, 50,000 Pa), and angles of attack were evaluated together [15]. The most favorable results in terms of aerodynamic efficiency (E_max) were obtained at a 45° blowing angle with maximum pressure. However, it was concluded that blowing pressure had a more dominant impact than angle, with a 5% improvement observed in lateral autonomous control performance compared to the baseline configuration. Finally, in a separate study addressing the effect of blowing pressure, four rectangular blowing regions (area: 0.0058 m²) were positioned at 0.6x/c chordwise and 0.2b and 0.6b spanwise locations on each wing. Among the tested blowing pressures (101,325 Pa, 202,650 Pa, and 303,975 Pa), the configuration operating at the lowest pressure yielded the most efficient result, with a 15% increase in aerodynamic efficiency [16].

Boutoudj ve Tebbiche conducted an experimental study in which they compared the effectiveness of vortex generators and micro-blowing techniques for controlling boundary layer separation over a NACA 0015 airfoil [17]. Triangular vortex generators were placed on the suction surface at a location corresponding to 10% of the chord length. Subsequently, this method was replaced with a series of micro-blowing holes, each 0.6 mm in diameter, arranged at regular intervals along the same chordwise position and oriented at an angle of 45 degrees to the chord line. Using the micro-blowing technique, they achieved a 49% increase in lift and a 69% reduction in drag. Xie et al. Investigated the effects of micro blowing and suction flow control devices on a NACA 0015 airfoil [18]. When the devices were placed at 12% and 30% chord positions, suction was found to be sometimes more effective than blowing, while at the 70% position both methods exhibited similar effects. The blowing/suction devices positioned near the separation point demonstrated higher efficiency, attributed to blowing enhancing turbulent kinetic energy by mixing jet and incoming flows, and suction mitigating low turbulent kinetic energy flow, thereby reducing separation. Truong et al. [19] demonstrated that optimal placement of Zero Net Mass Flux (ZNMF) actuators on a tilt rotor VTOL aircraft with a tilted nacelle during takeoff and landing effectively delays flow separation under near-stall conditions, resulting in significant reduction of pressure drag and enhancement of aerodynamic performance. Siliang et al [19]. conducted a study on a novel distributed jet blowing wing design for Fan-wing aircraft, based on the vortex-induced lift and thrust principle of the Fan-wing [20]. Numerical analyses demonstrated that the distributed jet blowing wing exhibits vortex-induced lift and thrust characteristics comparable to those of the Fan-wing, indicating that this technology can potentially replace the Fan-wing and be applied in ultrashort take-off and landing (USTOL) aircraft concepts. Within the scope of the study conducted by Sabırlı, both numerical and experimental investigations were carried out on wing models based on the NACA 5315 airfoil by applying vortex generators and steady blowing flow control techniques, both individually and in combination, across a range of freestream velocities and Reynolds numbers [21]. The vortex generators were positioned at 25% of the chord length from the leading edge, while the blowing holes were located at 60%. Due to their proximity to the leading edge, the vortex generators were observed to have a more pronounced effect. The implemented methods resulted in a delay of the stall angle by up to 6°, and an increase in the maximum lift coefficient by up to 20%. Although previous studies have provided significant insights into the combined use of active and passive flow control techniques, a comprehensive investigation involving the simultaneous application of vortex generators and steady blowing on a three-dimensional VTOL wing configuration is still lacking in the literature. The present study aims to fill this gap by offering a detailed and systematic analysis within this framework.

VTOL UAV Design

The first stage of the study focused on the design of a vertical take-off and landing (VTOL) unmanned aerial vehicle using a program capable of three-dimensional modeling. An airfoil with characteristics similar to those of the Bayraktar KALKAN DİHA was sought. Based on observations, the maximum thickness was approximately 30–40%, with a thickness ratio of 8.5–12% in the wing center section and about 4.3–5.4% in the rest of the wing. Considering that the lower surface of the wing was assumed to resemble a flat plate, the Fage & Collins 2 airfoil (fg2-il) was selected as the most suitable candidate according to the research conducted in airfoil databases. The maximum thickness of the Fage & Collins 2 airfoil is at 30% of the chord from the leading edge and the maximum thickness value is 8.2%. The maximum camber is also at 30% of the chord from the leading edge and the maximum camber value is 3.3%. Figure 2 provides a detailed illustration of the Fage & Collins 2 airfoil geometry.

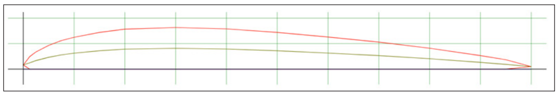

Figure 2: Fage & Collins 2 airfoil geometry [22]

At this stage, the wing model has been designed using a single airfoil profile. In subsequent phases, different airfoil profiles will be employed at various wing sections, and the outcomes will be comparatively evaluated. The wing model developed in this study is illustrated in Figure 3 and Table 1 summarizes the design parameters of the wing model.

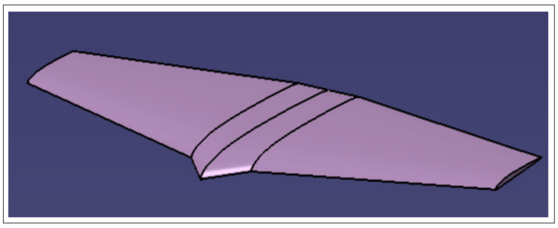

Figure 3: VTOL UAV wing model

Table 1: Wing model design parameters

Wing span:	5 m
Root chord length:	1.17 m
Junction section chord length:	0.98 m
Tip chord length:	0.42 m
Leading edge sweep angle:	10°
Trailing edge sweep angle:	-5°

Since a relatively thin airfoil is desired for the propeller to be used in the VTOL system, the ONERA HOR04 (HOR04-il) airfoil was selected at the initial stage from an airfoil database under the category of propeller blade airfoils. This airfoil has a maximum thickness of 4.1% located at 33.2% of the chord length from the leading edge. In the propeller design, the airfoil sections were positioned from the tip toward the root with angles of attack increasing by 0.5° increments, reaching up to a maximum angle of attack of 5°. The designed propeller is illustrated in Figure 4. In the VTOL configuration, a total of five propellers is planned to be used: four mounted on pods beneath the wings and one positioned behind the wing. Modifications to the propeller design are subject to change based on the results of the numerical analyses. Within the scope of this paper, numerical analysis is performed solely on the wing model.

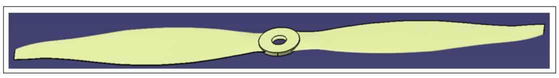

Figure 4: Propeller Model

Numerical Methods

To investigate the aerodynamic effects of the flow control techniques to be implemented in this study, computational fluid dynamics (CFD) simulations will be performed using ANSYS FLUENT. As a prerequisite, the development of an optimized computational mesh is essential. To determine the most accurate mesh configuration, a mesh validation study was conducted. Given the lack of experimental data in the literature concerning three-dimensional configurations based on the Fage & Collins

2 airfoil, the experimental results obtained by Boutoudj and Tebbiche [17] for a wing constructed using the NACA 0015 airfoil were employed as reference data for mesh validation. This wing model has a chord length of 15 cm and a wingspan of 20 cm.

Design and Validation of the Optimal Mesh Configuration

As a result of the literature review, it has been determined that when conducting a three-dimensional flow analysis, it is ideal to design the control volume such that the distances from the front, bottom, top, and sides of the body are 5 to 10 times the characteristic length of the body (mean chord length for a wing), and 15 to 20 times this length at the rear. In this context, for the wing model used in the validation study, a cylindrical enclosure domain with a hemispherical inlet was employed. The control volume was designed with a downstream length of twenty times the chord and an upstream hemispherical inlet with a radius approximately six times the chord length. The computational mesh was generated accordingly. Subsequently, the angle of attack of the model was increased from 0 to 20 degrees in two or four-degree increments, yielding a total of 8 different angles. For each case, the lift and drag coefficients were obtained at a free-stream velocity of 20 m/s and compared with experimental data. With these adjustments, the results remained consistent, leading to the decision to apply the developed mesh configuration to the wing model designed within the scope of this study. Visual representations of the validated wing model and the developed computational mesh are presented in Figure 5 and Figure 6 respectively.

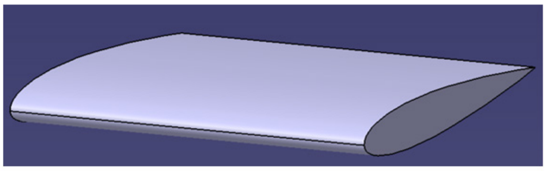

Figure 5: The wing model designed for validation purposes

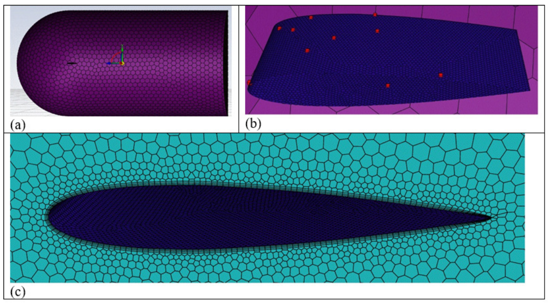

Figure 6: (a) 3D Volume Mesh of the Computational Domain, (b) Detailed Surface Mesh Distribution over the NACA0015 Wing Geometry, (c) Cross-Sectional View Showing Inflation Layers for Boundary Layer Resolution

The control volume specifications are summarized in Table 2, and the mesh-related information is given in Table 3. Additionally, the parameters used for the computational fluid dynamics (CFD) analysis are provided in Table 4.

Table 2: Geometrical and boundary features of the domain used in mesh validation

Control Volume Characteristics	
Domain Shape	Cylindrical Enclosure with
Hemispherical Inlet	0.98 m
Radius:	1 m
Upstream Distance	1 m
Downstream Distance	3 m
Total Volume	5.73 m3
Boundary Conditions	Velocity Inlet, Pressure outlet, Symmetry Plane, Interior, Wall (Wing Surface)

Table 3: Mesh characteristics generated for the purpose of validation

Mesh Charecteristics	
Mesh Type	Polyhedral
Surface Mesh Type	Polygonal
Total Number of Cells	164572 polyhedra cell
Maksimum Skewness	0.7
Mesh Quality (Orthogonality)	0.3
Minimum Face Size	0,0019 m
Maximum Face Size	0,099 m
Inflation Layers	3
Growth Rate	1.2
Meshing Tool	Fluent Meshing

Table 4: Computational fluid dynamics (CFD) analysis parameters

Analysis Parameters	
Flow Type	Turbulent k-ω -SST
Density Model	Incompressible
Time Approach	Steady
Reynolds Number	266000
Solver Type	Pressure Based
Boundary Condition	Velocity Inlet

Numerical Investigations of a Baseline Wing Model

The validated mesh was also applied to the baseline wing model, and numerical analyses were conducted accordingly. The results obtained from these simulations will be presented in the Results and Discussion section at a later stage.

Numerical Evaluation of a Wing Model Equipped with Flow Control Techniques

Following the conducted analyses, it was determined that low-profile vortex generators are more effective in suppressing flow separation. Therefore, in the initial phase of the study, the impact of this type of vortex generators on aerodynamic performance was investigated. In this context, 31 counter-rotating vortex generators were placed on the wing model, corresponding to 20% of the chord length from the leading edge (see Figure 7). The horizontal spacing between the centers of the vortex generators was set to 16 cm. Each vortex generator was designed with a length and depth of 3 cm. The heights of the vortex generators will be determined in accordance with the boundary layer thickness derived from the post-processed flow analysis data.

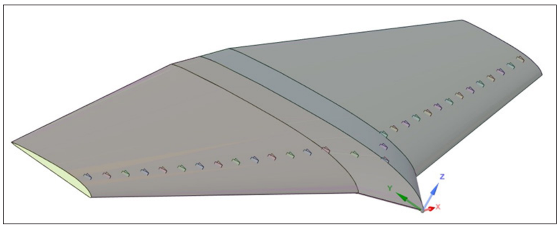

Figure 7: Wing model equipped with vortex generators

In light of the investigations conducted in the literature, it has been observed that the parameters of the blowing system have a significant effect on flight performance. When the blowing system variables are optimized collectively and applied in an integrated manner, the aerodynamic efficiency of the unmanned aerial vehicle (UAV) increases considerably. Therefore, it would be more beneficial to consider all variables holistically rather than individually.

According to findings in the literature, positioning in the chordwise direction is more effective than positioning in the spanwise direction. When the blowing location is positioned closer to the leading edge in the chordwise direction, the aerodynamic effectiveness of the system increases. However, beyond a certain point, flow separation occurs due to adverse flow induced by the presence of the blowing system. This is particularly evident when the blowing system is located near the camber region of the airfoil. In fact, the point where reverse flow occurs varies depending on factors such as the geometry of the blowing region and the magnitude of the blowing pressure outlet. Therefore, in determining the optimal location of the blowing system, it is crucial to consider the UAV's maximum lift coefficient to prevent flow separation at low angles of attack [2]. In spanwise positioning, placing the blowing system near the wingtip has been found to be more efficient. The underlying reason is the increasing influence of shear effects along the wing span, resulting in more intense vortex formation near the wingtip.

When evaluating the blowing area, both shape and size should be considered separately. In terms of length, although a blowing system extending along the entire wingspan provides greater performance improvement, it may present structural disadvantages. As the blowing area increases, delaying flow separation becomes easier, but the required amount of pressurized air also increases. Consequently, a larger compressor is needed, which leads to an increase in overall weight. There are two commonly used blowing outlet shapes: circular and rectangular. Blowing regions are typically positioned at or just upstream of the flow separation onset. Rectangular blowing outlets are particularly advantageous when the separation point is known for a specific angle of attack, whereas circular outlets are more effective when the separation onset shifts with varying angles of attack.

Ideally, the blowing pressure magnitude should be kept as low as possible. Increasing the pressure of the blown air requires a larger pressurized air source, which is undesirable from a structural standpoint. However, with an optimal combination of blowing parameters, performance improvements can still be achieved even at relatively low blowing pressure magnitudes.

When the blowing outlet angle is considered as the sole variable, it does not have a significant impact on aerodynamic efficiency. Its effect becomes more pronounced when evaluated together with other variables. Among the angles studied, 45° has been found to be the most efficient.

In the initial stage, the blowing system to be tested for its effectiveness is designed with parameters including a 45° blowing angle, a blowing pressure of 101,325 Pa, a circular cross-section, and a blowing area with a diameter of 2 mm. The positioning of this system is planned to be determined following the baseline wing analysis. If flow separation is observed, the blowing units will be located just upstream of the identified separation points.

Result and Discussion

This paper presents the findings of an ongoing, broader research project at its current stage. Upon completion, the study is expected to contribute more extensive results to the literature.

For the purpose of mesh verification, the lift and drag coefficient graphs provided by Boutoudj and Tebbiche at various Reynolds numbers will be employed (see Figure 8). Given that the validation analysis is conducted at a flow velocity of 20 m/s—corresponding to a Reynolds number of approximately 266,000—it is deemed sufficient that the obtained lift and drag coefficient values exhibit convergence toward those associated with a Reynolds number of 250,000.

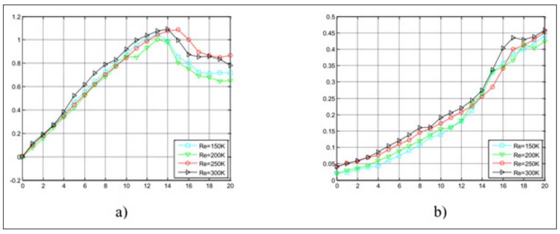

Figure 8: Lift and drag coefficients versus angles of attack at various Reynolds numbers.
a) Lift coefficient
b) Drag coefficient [17].

The lift and drag coefficient values obtained from the computational fluid dynamics (CFD) analysis conducted in the Fluent were compared with the experimental data, as illustrated in Figure 9. The degree of agreement between the results supports the applicability of the computational mesh. In the subsequent studies, the mesh will be further refined based on the same mesh parameters.

For the reference wing model used in the validation study, the static pressure, velocity magnitude, and turbulent kinetic energy contours obtained at an angle of attack of 16° are illustrated in Figure 10.

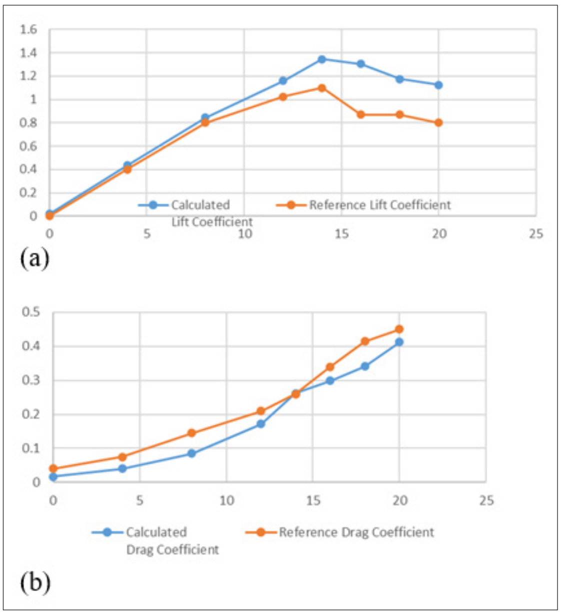

Figure 9: (a) Comparison of Lift Coefficient Values (b) Comparison of Drag Coefficient Values

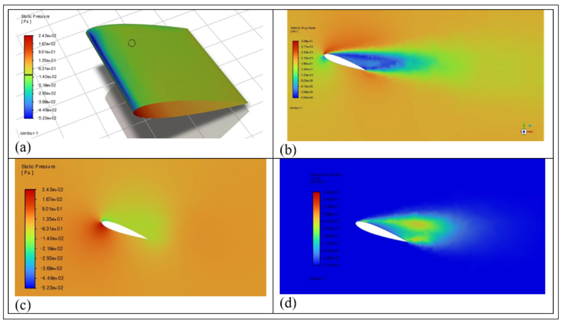

Figure 10: (a) Static Pressure distribution for NACA0015 wing model, and respectively, (b) Velocity Magnitude contour, (c) Static Pressure Contour, (d) Turbulent Kinetic Energy Contour for NACA0015 Wing Section

References

1. Çobanoğlu T, Taşçıoğlu R, Şahin S, Sevinç N, Yazanel M. Experimental Investigation of Active and Passive Flow Control Systems Using Smoky Flow Imaging Technique. 2022.
2. Wing PTO. Application of Active Flow Control and Autonomous Control Methods on Unmanned Aerial Vehicles. PhD Thesis, Erciyes University, Kayseri. 2020.
3. Yazanel M. Effect of Active and Passive Flows on Airfoil In: Kahraman N, Aslan A, Soylak M, Soytürk MA, Yıldız M (eds) Proceedings of the 1st Symposium of the Faculty of Aerospace Sciences (I. HUBF). Kayseri. 2024. 52-61.
4. Abbas A, De Vicente J, Valero E. Aerodynamic technologies to improve aircraft performance. Elsevier. 2012. 28: 100-132.
5. Yıldız Aydemir E. Flow Control Methods Applied to Unmanned Aerial Vehicles and Their Effects on UAV Performance. In: Yıldız M, Yavuz S (eds) Proceedings of the 2nd Symposium of the Faculty of Aerospace Sciences (II. HUBF). Kayseri. 2024. 167-180.
6. Liu Y, Wu M, Zhu JJ, Lawrence DA, Gutmark EJ, et al. Reactive flow control of delta-wing vortex. Journal of Aircraft. 2008. 45: 880-892.
7. Koike S, Nakakita K, Nakajima T, Koga S, Sato M, et al. Experimental investigation of vortex generator effect on two- and three-dimensional NASA common research models. In: 53rd AIAA Aerospace Sciences Meeting. AIAA, Kissimmee, Florida. 2015.
8. Lin JC. Review of research on low-profile vortex generators to control boundary-layer separation. Prog Aerosp Sci. 2002. 38: 389-420.
9. Lin JC. Control of turbulent boundary-layer separation using micro-vortex generators. In: 30th AIAA Fluid Dynamics Conference. AIAA, Norfolk, VA. 1999.
10. Oktay T, Uzun M, Kanat ÖÖ. Improvement of wingtip vortex with active flow control technique. In: Proceedings of the 2nd International Defense Industry Symposium (IDEFIS 2017), Kırıkkale, Turkey. 2017.
11. Oktay T, Kanat ÖÖ, Ceyhan A. Improvement of wingtip vortices with active flow controller having several blowing points. Proceedings of the 8th International Advanced Technologies Symposium (IATS 2017), Elazığ, Turkey. 2017.
12. Kanat ÖÖ, Oktay T. Effect of blower location and radius on lateral flight control system of UAVs. In: Proceedings of the International Congress of Multidisciplinary Studies. 2018.
13. Kanat ÖÖ, Oktay T. Effect of blower location and radius on longitudinal flight control system of UAVs. In: Proceedings of the International Congress of Multidisciplinary Studies. 2018.
14. Oktay T, Kanat ÖÖ. The importance of blowing angle while designing an active blowing system. In: 5th International Conference on Engineering and Natural Science. Prague, Czech Republic. 2019.
15. Oktay T, Kanat ÖÖ. The aerodynamic effects of pressure magnitude and blowing angle on lateral autonomous performance of UAVs. In: 5th International Conference on Engineering and Natural Science. Prague, Czech Republic. 2019.
16. Oktay T, Uzun M, Kanat ÖÖ. Maximum lift/drag ratio improvement of TUAVs via small aerodynamic modifications. Aircr Eng Aerosp Technol. 2018. 90:1438–1444.
17. Boutoudj MS, Tebbiche H. Effects of micro-blowing and vortex generators on the boundary layer separation control of a NACA 0015 airfoil. Technol Eng Math (EPSTEM). 2024. 32.
18. Xie L, Zheng Y, Yao Y. Effects of Micro Blowing/Suction Flow Control Devices on a NACA0015 Airfoil Computational Fluid Dynamics. In: Zhang HX, Deng XG, Fan ZL (eds) The Eighth International Conference on Computational Fluid Dynamics (ICCFD8). Chengdu, China. 2014. 702-718.
19. Truong H, Marouf A, Vos JB, Gehri A, Hoarau Y. Aerodynamic drag reduction of a tilt rotor aircraft using zero-net-mass-flux. 2023.
20. Siliang D, Qijun Z, Bo W. Research on Distributed Jet Blowing Wing Based on the Principle of Fan-Wing Vortex-Induced Lift and Thrust. Int J Aerosp Eng. 2019.

21. Sabırlı MÜ. Improvement of Wing Aerodynamic Performance Parameters by Active/Passive Flow Control on Wings. PhD Thesis, Bursa Uludağ University, Bursa. 2025.
22. Airfoil Tools. http://airfoiltools.com/