

CFD Investigation of Flow Over Grooved Surface

Rohit Kadam¹, Harshada Gaikwad², Somesh Naikwadi³, Sachin Chavan³, Pratik Gaikwad³,

Anmol Rayrikar³

¹Department of Aeronautical And Aerospace Engineering Sanjay Ghodawat University Kolhapur

²Department of Computational Mechanics PCCOE Pune

³Department of Mechanical Engineering TSSM's Padmabhooshan Vasant dada Patil institute of technology, Bavdhan, Pune

Abstract - Flow separation over surfaces is a critical issue in aerodynamic and fluid flow applications, leading to increased drag and reduced efficiency. To address this, surface modification techniques such as grooved geometries have been explored for passive flow control. This study focuses on the computational analysis of jet flow over grooved surfaces using Computational Fluid Dynamics (CFD). The research aims to investigate the influence of different groove shapes circular, and triangular on flow behaviour, vortex formation, and jet-surface interaction. A computational domain is developed, and simulations are conducted with appropriate meshing, boundary conditions, and solver configurations. Special emphasis is placed on evaluating the effect of standoff distance on jet characteristics. The study also addresses existing research gaps, including limited comparative analysis of groove geometries and insufficient understanding of vortex strength and velocity distribution near grooved surfaces.

The expected outcomes include identifying the most effective groove configuration for enhanced flow control and improved aerodynamic performance. The findings of this research can contribute to applications in aircraft surface design and jet flow management systems.

Key Words: Grooved Surface, Passive Flow Control, Jet Flow, Flow Separation Control, Boundary Layer Separation, Drag Reduction, Vortex Dynamics, Aerodynamics, Riblet, Standoff Distance, Flow Control, Reynold Number, Angle of Attack.

1. INTRODUCTION

Airflow over surfaces often leads to flow separation, which results in increased drag and reduced efficiency. To overcome this, engineers apply various flow control techniques to improve aerodynamic performance. One effective passive method is the use of grooved surfaces, which help in modifying near-wall flow behavior and controlling separation. Smooth surfaces cannot effectively control flow separation. There is limited understanding of how different groove shapes affect jet flow.

2. MOTIVATION

Grooved surfaces (riblets) are widely studied as a passive flow control technique for improving aerodynamic performance and reducing drag. Their development is inspired by both natural surfaces such as shark skin and engineering devices like vortex generators, which are used to control boundary layer behaviour and delay flow separation. These micro-scale grooves

modify near-wall turbulence and influence vortex motion within the boundary layer.

With the advancement of Computational Fluid Dynamics (CFD), detailed analysis of flow characteristics such as velocity distribution, turbulence structures, and pressure variations over such modified surfaces has become possible. Most existing studies focus on flow over flat plates, airfoils, and channels with grooved geometries.

However, limited research has been carried out on the interaction between jet flow and grooved surfaces, which is important in applications such as flow control, cooling, and propulsion systems. Understanding this interaction is essential to evaluate how grooves influence jet-induced flow behaviour and vortex dynamics.

Therefore, the proposed research focuses on the CFD analysis of jet flow over grooved surfaces to better understand the underlying flow mechanisms and potential performance improvements.



Fig -1: Vortex Generators

3. FLOW MANIPULATION TECHNIQUES OVER BODIES FOR AERODYNAMIC BENEFITS

Three forms of flow control methodologies are focusing on controlling flow separation in order to minimize the effect of drag on the aerodynamic body as Design optimization in lifting surfaces, smoothening flow through design changes of sharp edges, and flow control devices [1].

Over the few past decades, there has been a significant advancement in flow control studies and related applications. The goal of this rapidly expanding, multidisciplinary scientific subject is to transform a natural flow condition into one that is more efficient. Transition delay, drag reduction, mixing augmentation, lift enhancement, mixing augmentation, flow-induced noise suppression, turbulence management, and separation postponement, are the main goals of contemporary flow control techniques [2].

There is still room for improvement in weight, fabrication cost, cruising efficiency, and part count even if the high-lift system on contemporary transport aircrafts are far simpler than they were in the past [3].

These methods involve making geometrical adjustments to the engine nozzle and lifting surfaces. Moreover, a number of flow control techniques are used to improve aircraft performance even more. Techniques for both passive & active flow control are used under typical flight circumstances [4].

3.1 ACTIVE FLOW

In recent years, significant research has focused on Active Flow Control (AFC) techniques to improve aerodynamic performance by delaying flow separation and increasing lift [4]. These methods use fluidic actuation, often supplied by the engine, to reduce or eliminate flow separation, resulting in improved lift and reduced momentum requirements [4]. Studies have shown that optimal excitation frequencies identified using Fast Fourier Transform (FFT) can enhance lift performance by up to 8%, making modified airfoils such as slatted airfoils more efficient than conventional designs [5][6]. In addition, morphing wing technologies such as Morphing Trailing Edge (MTE) help reduce drag and improve fuel efficiency, where negative twist angles reduce drag and higher positive twist angles may decrease aerodynamic performance [8][9]. Overall, these advanced techniques contribute to better lift, lower drag, and improved aircraft efficiency.

3.2 PASSIVE FLOW

In recent years, passive flow control techniques have gained attention as effective methods to improve aerodynamic performance without requiring external energy input. These techniques modify the surface geometry to control boundary layer behaviour and delay flow separation, thereby reducing drag and improving efficiency. Common passive methods include vortex generators, riblets, and grooved surfaces, which influence vortex formation and enhance momentum transfer near the surface. Studies have shown that such surface modifications can significantly reduce skin friction and improve flow stability under various operating conditions. In particular, different groove geometries such as rectangular, circular, and triangular shapes affect vortex strength and flow interaction differently, leading to variations in performance. Additionally, parameters like standoff distance play an important role in determining jet-surface interaction and overall flow behaviour. Despite their advantages, there is limited comparative research on different groove configurations and their combined effects on flow characteristics. Overall, passive flow control techniques offer a simple, energy-efficient, and reliable approach for improving aerodynamic performance in applications such as aircraft surfaces and fluid flow systems [1][2][3].

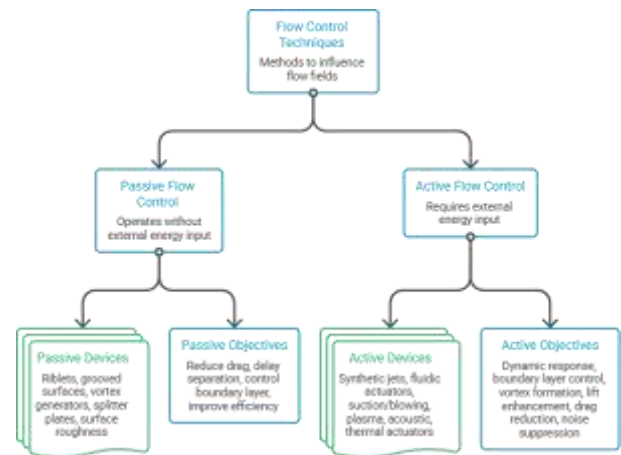


Fig -2: Flow Manipulation Techniques Over Bodies For Aerodynamic Benefits

4. OBSERVATIONS AND GAPS IN LITERATURE

The existing literature on jet flow over surfaces highlights the benefits of passive flow control techniques such as grooved geometries; however, several important gaps remain. Most studies have focused on limited or single groove configurations, leading to a lack of comprehensive comparative analysis between different groove shapes such as rectangular, circular, and triangular profiles. Additionally, the influence of standoff distance (h/De) on jet-surface interaction has not been explored in sufficient detail, particularly in relation to its impact on flow separation control. There is also an incomplete understanding of the underlying flow physics, especially concerning vortex formation, vortex strength, and velocity distribution in the vicinity of grooves. Furthermore, many studies rely on simplified assumptions or lack detailed CFD-based investigations that capture these complex interactions. Therefore, a systematic numerical study addressing these parameters is necessary to bridge the gap and improve the effectiveness of groove-based flow control strategies.

5. OBJECTIVE OF THE PROPOSED RESEARCH

The objective of the proposed research is to investigate the effect of grooved surface geometries on jet flow behavior and their potential in controlling flow separation. The study aims to perform a detailed Computational Fluid Dynamics (CFD) analysis to evaluate how different groove shapes influence velocity distribution, vortex formation, and overall aerodynamic performance. Another key objective is to examine the role of standoff distance in modifying jet-surface interactions and its impact on flow control effectiveness. Inspired by vortex generator concepts, the research seeks to understand how passive surface modifications can enhance flow attachment and reduce aerodynamic losses. Ultimately, the goal is to develop an optimized grooved surface configuration that improves flow characteristics and contributes to increased efficiency in

aerodynamic

applications.

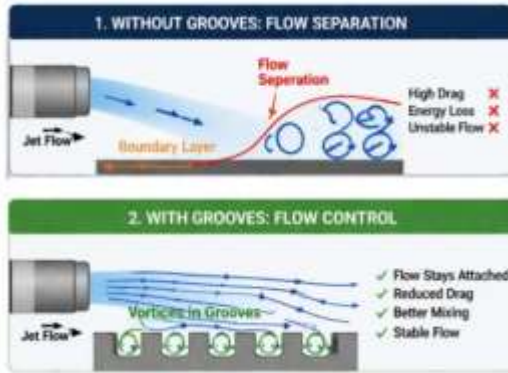


Fig -3: Comparison of flow behavior with and without grooves

6. METHODOLOGY

The methodology for the proposed research involves a systematic Computational Fluid Dynamics (CFD) approach to analyze jet flow over grooved surfaces. Initially, a baseline smooth surface model will be created, followed by the design of different grooved geometries such as rectangular, circular, and triangular profiles. These models will be developed using suitable CAD software and then imported into a CFD tool for analysis. A structured or hybrid mesh will be generated with proper refinement near the wall and groove regions to accurately capture boundary layer effects and vortex formation. Boundary conditions will be defined by specifying jet inlet velocity, outlet pressure conditions, and appropriate turbulence models to simulate realistic flow behavior. Simulations will be carried out for varying standoff distances to study jet-surface interaction in detail. The results will be analyzed in terms of velocity contours, pressure distribution, and vortex structures to understand flow characteristics. A comparative evaluation between different groove configurations and the smooth surface will be performed to assess improvements in flow attachment and reduction in separation. Finally, conclusions will be drawn to identify the most effective groove design for enhanced aerodynamic performance.

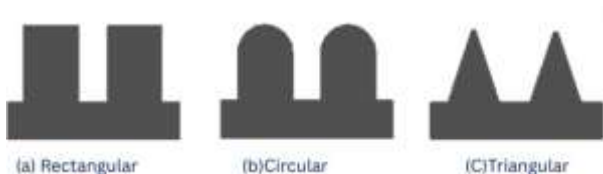


Fig -4: Different proposed grooves

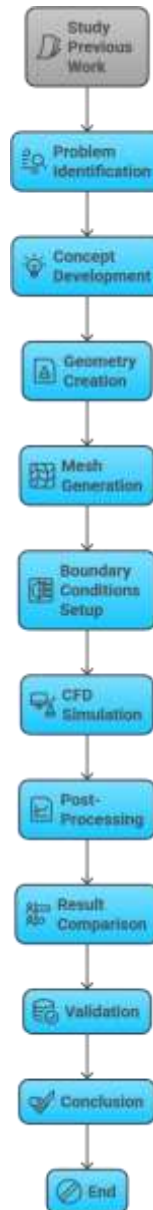


Fig -5: Flowchart of The Methodology for the Proposed Research Work

7. CFD Simulations Using Ansys Fluent

A 2D geometric model with proposed groove structures was successfully developed using ANSYS Design Modeler. The incorporation of periodic grooves enables detailed investigation of their effect on fluid flow behavior, particularly in reducing skin friction drag. The prepared geometry serves as a robust foundation for further CFD analysis and optimization studies. A two-dimensional computational domain was created in ANSYS Workbench using ANSYS Design Modeler, consisting of an inlet, test section, and outlet region. Periodic groove structures (riblets) were introduced on the bottom wall by first sketching a single groove element and then applying a linear pattern to maintain uniform spacing along the surface. The geometry was then meshed using a structured approach, with finer elements near the grooved region to accurately capture boundary layer effects and coarser mesh in the remaining domain to optimize

computational cost. Appropriate boundary conditions were applied, including a velocity inlet, pressure outlet, and no-slip wall condition. The simulation was performed using a steady-state solver until convergence was achieved. Post-processing of results included velocity and static pressure contours along with path lines, which showed the influence of grooves on near-wall flow behavior, indicating changes in velocity distribution and pressure characteristics relevant to drag reduction analysis.

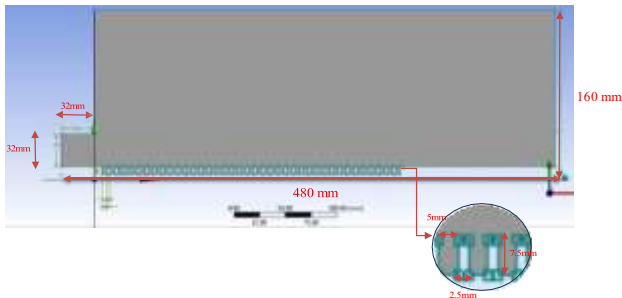


Fig -6: Model With proposed Grooves

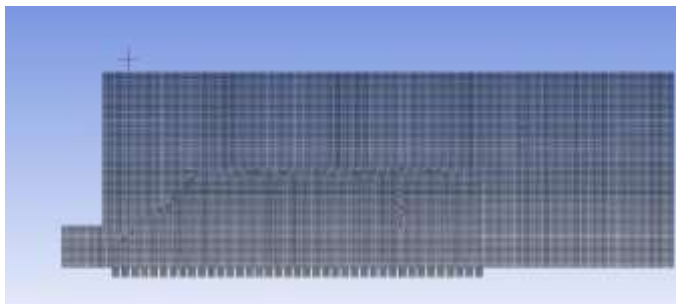


Fig -7: Structured Meshing of Proposed Grooved Model

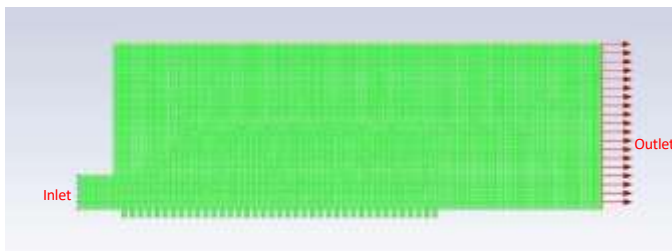


Fig -8: Applied Boundary Conditions

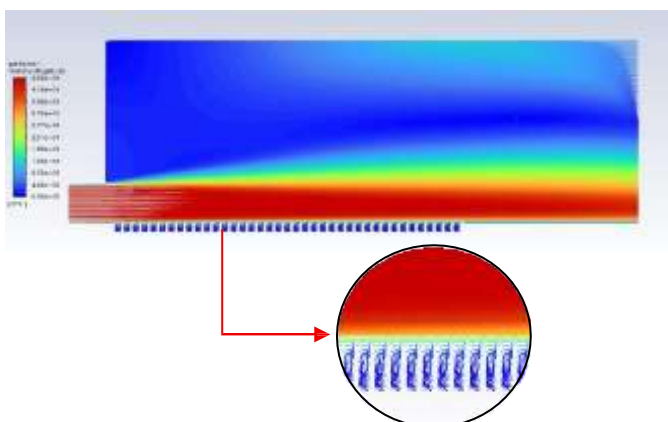


Fig -9: Velocity Contour

Boundary Type	Parameter	Value / Setting	Description
Inlet	Type	Velocity Inlet	Defines incoming flow
	Velocity	45m/s	
	Temperature	300 K	Standard ambient condition
Outlet	Type	Pressure Outlet	Allows flow to exit
	Gauge Pressure	0 Pa	Atmospheric reference
	Backflow Turbulence	5%	Stability during reverse flow
Wall	Type	No-slip Condition	Velocity = 0 at wall
	Roughness	Smooth (or as per grooves)	Defines wall interaction
	Heat Transfer	Adiabatic	No heat transfer assumed
Solver	Solver Type	Pressure-based, Steady	Suitable for incompressible flow
	Turbulence Model	k- ω SST	Accurate near-wall modelling
	Discretization	Second-order	Improves accuracy
Fluid Properties	Convergence Criteria	Residuals $< 10^{-5}$	Solution accuracy
	Fluid	Air	Working fluid
	Density	1.225 kg/m ³	At standard conditions
	Viscosity	1.789 $\times 10^{-5}$ kg/m·s	Dynamic viscosity

Table 1: Applied Boundary Conditions

8. Comparative CFD Analysis of Groove Geometries at Different Standoff Distances

At 0 standoff distance, the rectangular groove shows a higher pressure drop compared to the plain surface, indicating increased flow resistance due to surface modification.

└The velocity in the rectangular case is slightly higher than the plain case, suggesting localized acceleration of flow near the grooved region.

└Turbulence is significantly higher in the rectangular geometry, which indicates stronger disturbance and mixing caused by the grooves, whereas the plain surface maintains relatively smoother flow.

Geometry	Nodes	Elements	Pressure (Pa)	Velocity (m/s)	Turbulence (m ² /s ²)	Convergence
Plain	18497	18176	-8.0135	14.22561	21.56999	3491 (Converged)
Rectangular	18566	18093	-9.71022	14.32486	25.6858	4044 (Converged)

Table 2: Results at 0 Standoff Distance

└At 0.12 standoff distance, the plain surface shows more uniform velocity distribution, while the rectangular case exhibits reduced velocity, indicating energy loss due to groove interaction.

└The pressure drop remains higher in the rectangular case, confirming increased drag effects.

└Turbulence levels in both cases are similar, but the rectangular surface still shows slightly higher turbulence, indicating continued flow disturbance.

Geometry	Nodes	Elements	Pressure (Pa)	Velocity (m/s)	Turbulence (m ² /s ²)	Convergence
Plain	18405	18174	-22.7249	13.95357	21.18989	3806 (Converged)
Rectangular	18503	18031	-22.3175	12.67199	21.19544	4403 (Converged)

Table 3: Results at 0.12 Standoff Distance

└At 0.3 standoff distance, the rectangular geometry shows a much higher pressure drop than the plain surface, highlighting stronger resistance at increased distance.

└The velocity values are nearly similar, but contours indicate that the rectangular case has more disturbed flow patterns, especially near the wall region.

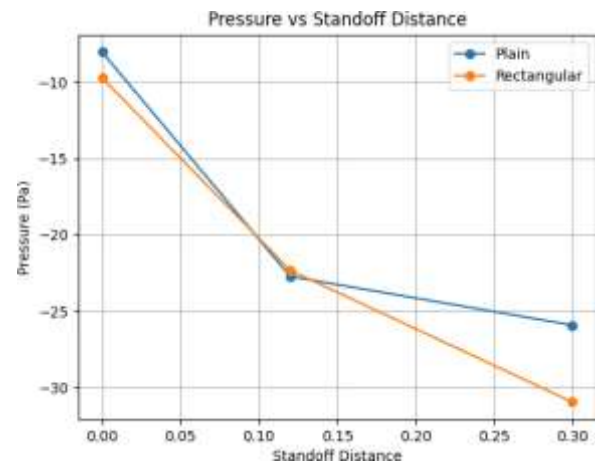
└Turbulence is again higher in the rectangular case, suggesting persistent vortices and mixing effects compared to smoother flow over the plain surface.

Geometry	Nodes	Elements	Pressure (Pa)	Velocity (m/s)	Turbulence (m ² /s ²)	Convergence
Plain	18492	18171	-25.9137	13.88323	21.70463	3868 (converged)
Rectangular	18549	18075	-30.9557	13.92684	24.80161	3860 (converged)

Table 4: Results at 0.12 Standoff Distance

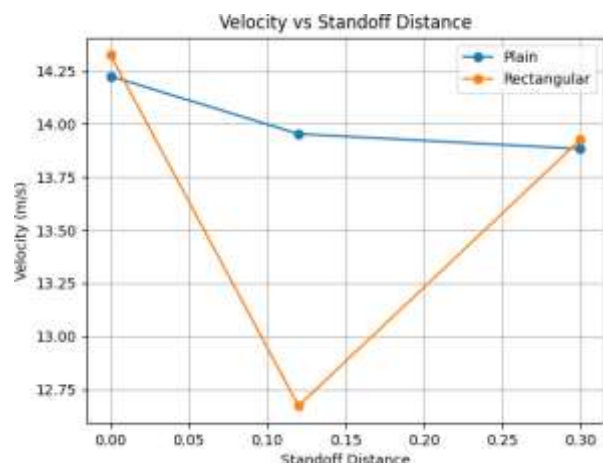
9. Pressure Vs Standoff Distance Plot

The pressure graph shows that both geometries experience an increase in pressure drop (more negative values) as standoff distance increases. However, the rectangular geometry consistently exhibits a higher pressure drop compared to the plain surface, especially at 0.3 standoff distance. This indicates that the rectangular grooves introduce additional flow resistance, leading to increased aerodynamic losses. From a performance perspective, the plain surface is more favorable in terms of lower drag.



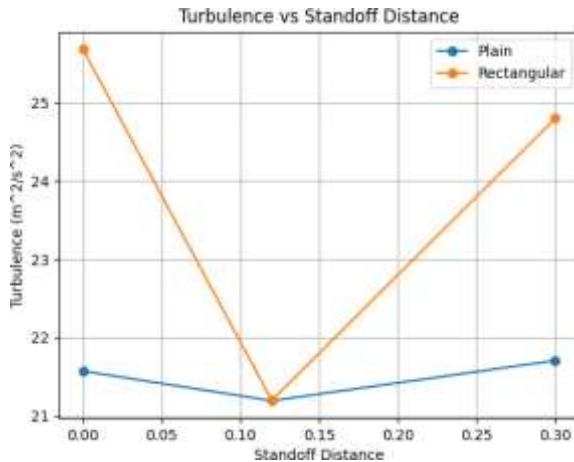
10. Velocity Vs Standoff Distance Plot

The velocity graph indicates that at zero standoff distance, the rectangular case shows a slightly higher velocity than the plain surface due to localized acceleration near grooves. However, at 0.12 standoff distance, the velocity in the rectangular case drops significantly compared to the plain case, suggesting energy loss due to flow interaction with grooves. At 0.3 standoff distance, both geometries show similar velocity values, but the rectangular case still reflects less uniform flow behavior. Overall, the plain geometry maintains more stable velocity characteristics.



11. Turbulence Vs Standoff Distance Plot

The turbulence graph clearly shows that the rectangular geometry generates higher turbulence levels compared to the plain surface across all standoff distances. This is due to the formation of small vortices and recirculation zones within the grooves. While increased turbulence can enhance mixing, it also leads to higher energy dissipation and drag. In contrast, the plain surface exhibits relatively lower and more stable turbulence, indicating smoother flow conditions.



12. CONCLUSIONS

The present study investigated the effect of rectangular groove structures on flow behavior by comparing them with a plain surface using CFD analysis in ANSYS Workbench. From the results obtained through pressure, velocity, and turbulence analysis, it is observed that the rectangular grooved surface increases pressure drop and turbulence intensity across all standoff distances. Although slight variations in velocity were noted, they were not sufficient to compensate for the increased energy losses caused by flow disturbances and recirculation within the grooves. In contrast, the plain surface demonstrated smoother flow characteristics, lower turbulence, and reduced pressure losses, indicating better aerodynamic efficiency. Therefore, the proposed rectangular groove design, under the given conditions, does not contribute to drag reduction and instead leads to higher flow resistance.

However, this study establishes a foundation for future research, where different groove geometries such as triangular, circular, or optimized riblet structures can be implemented and analyzed. With proper design and scaling, such surface modifications have the potential to control boundary layer behavior and delay flow separation, which is highly beneficial in aerodynamic applications like aircraft surfaces. Thus, while the current configuration did not yield performance improvement, it provides valuable insight and a basis for further investigation into groove-based passive flow control techniques.

ACKNOWLEDGEMENT

We express our sincere gratitude to everyone who contributed directly and indirectly to the successful completion of this project.

We would like to express our heartfelt gratitude to Mr Sulthan S, Assistant Professor & Head, Department of Aerospace SGU Kolhapur, Prof. S. D. Thorbole, Department of Mechanical Engineering TSSM's PadmaBhooshan Vasant dada Patil Institute of Technology (PVPIT), Bavdhan Pune, for their valuable guidance, continuous encouragement, constructive suggestions, and unwavering support throughout the project. Their expertise and insights greatly helped us in completing this research successfully.

REFERENCES

- [1] P. R. Ashill, J. L. Fulker, and K. C. Hackett, "A review of recent developments in flow control," *Aeronaut. J.*, vol. 109, no. 1095, pp. 205–232, May 2005, doi: 10.1017/S0001924000005200.
- [2] A. M. Halawa, B. Elhadidi, and S. Yoshida, "Aerodynamic Performance Enhancement Using Active Flow Control on DU96-W-180 Wind Turbine Airfoil," *Evergreen*, vol. 5, no. 1, pp. 16–24, Mar. 2018, doi: 10.5109/1929723.
- [3] M. Desalvo, E. Whalen, and A. Glezer, "High-Lift Enhancement Using Active Flow Control," in 6th AIAA Flow Control Conference, New Orleans, Louisiana: American Institute of Aeronautics and Astronautics, Jun. 2012. doi: 10.2514/6.2012-3245.
- [4] Y. Yadlin and A. Shmilovich, "Lift Enhancement for Upper Surface Blowing Airplanes," in 31st AIAA Applied Aerodynamics Conference, San Diego, CA: American Institute of Aeronautics and Astronautics, Jun. 2013. doi: 10.2514/6.2013-2796.
- [5] M. DeSalvo, E. Whalen, and A. Glezer, "High-Lift Performance Enhancement Using Active Flow Control," *AIAA J.*, vol. 58, no. 10, pp. 4228–4242, Oct. 2020, doi: 10.2514/1.J059143.
- [6] A. M. Halawa, B. Elhadidi, and S. Yoshida, "Aerodynamic Performance Enhancement Using Active Flow Control on DU96-W-180 Wind Turbine Airfoil," *Evergreen*, vol. 5, no. 1, pp. 16–24, Mar. 2018, doi: 10.5109/1929723.
- [7] J. Zhang, K. Xu, Y. Yang, Y. Ren, P. Patel, and G. Zha, "Aircraft Control Surfaces Using Co-flow Jet Active Flow Control Airfoil," in 2018 Applied Aerodynamics Conference, Atlanta, Georgia: American Institute of Aeronautics and Astronautics, Jun. 2018. doi: 10.2514/6.2018-3067.
- [8] D. Communier, R. M. Botez, and T. Wong, "Experimental validation of a new morphing trailing edge system using Price – Païdoussis wind tunnel tests," *Chin. J. Aeronaut.*, vol. 32, no. 6, pp. 1353–1366, Jun. 2019, doi: 10.1016/j.cja.2019.03.016.
- [9] E. Kaygan and C. Ulusoy, "Effectiveness of Twist Morphing Wing on Aerodynamic Performance and Control of an Aircraft," *J. Aviat.*, vol. 2, no. 2, pp. 77–86, Dec. 2018, doi: 10.30518/jav.482507.
- [10] T. Xiao, Z. Zhu, S. Deng, F. Gui, Z. Li, and Z. Zhou, "Effects of nozzle geometry and active blowing on lift enhancement for upper surface blowing configuration," *Aerosp. Sci. Technol.*, vol. 111, p. 106536, Apr. 2021, doi: 10.1016/j.ast.2021.106536.
- [11] Y. Yuhui and Z. Yufei, "Optimization and Analysis of a Blown Flap Based on a Multielement Airfoil," *J. Aircr.*, vol. 57, no. 1, pp. 62–75, Jan. 2020, doi: 10.2514/1.C035514.
- [12] J. Sun and D. Huang, "Numerical investigation of boundary layer suction control positions on airfoils for vertical-axis wind turbine," *J. Mech. Sci. Technol.*, vol. 35, no. 7, pp. 2903–2914, Jul. 2021, doi: 10.1007/s12206-021-0614-0.
- [13] V. Kornilov, "Combined Blowing/Suction Flow Control on Low-Speed Airfoils," *Flow Turbul. Combust.*, vol. 106, no. 1, pp. 81–108, Jan. 2021, doi: 10.1007/s10494-020-00157-7.
- [14] M. Kotsonis, "Diagnostics for characterization of plasma actuators," *Meas. Sci. Technol.*, vol. 26, no. 9, p. 092001, Sep. 2015, doi: 10.1088/0957-0233/26/9/092001.
- [15] M. Kotsonis, R. Pul, and L. Veldhuis, "Influence of circulation on a rounded-trailing-edge airfoil using plasma actuators," *Exp. Fluids*, vol. 55, no. 7, p. 1772, Jul. 2014, doi: 10.1007/s00348-014-1772-y.
- [16] S. Cao, N. Dang, Z. Ren, J. Zhang, and Y. Deguchi, "Lagrangian analysis on routes to lift enhancement of airfoil by synthetic jet and their relationships with jet parameters," *Aerosp. Sci. Technol.*, vol. 104, p. 105947, Sep. 2020, doi: 10.1016/j.ast.2020.105947.
- [17] P. Zhang, B. Yan, and C. Dai, "Lift enhancement method by synthetic jet circulation control," *Sci. China Technol. Sci.*, vol. 55,

- no. 9, pp. 2585–2592, Sep. 2012, doi: 10.1007/s11431-012-4896-4.
- [18] G.-C. Zha and C. Paxton, “A Novel Airfoil Circulation Augment Flow Control Method Using Co-Flow Jet,” in 2nd AIAA Flow Control Conference, Portland, Oregon: American Institute of Aeronautics and Astronautics, Jun. 2004. doi: 10.2514/6.2004-2208.
- [19] S. Kumar, K. M. Pandey, and K. K. Sharma, “Advances in drag-reduction methods related with boundary layer control – A review,” *Mater. Today Proc.*, vol. 45, pp. 6694–6701, 2021, doi: 10.1016/j.matpr.2020.12.150.
- [20] B. Dano, D. Kirk, and G. Zha, “Experimental Investigation of Jet Mixing Mechanism of Co-Flow Jet Airfoil,” in 5th Flow Control Conference, Chicago, Illinois: American Institute of Aeronautics and Astronautics, Jun. 2010. doi: 10.2514/6.2010-4421.
- [21] R. Abinav, N. R. Nair, P. Sravan, P. Kumar, and S. R. Nagaraja, “CFD Analysis of Co Flow Jet Airfoil,” *Indian J. Sci. Technol.*, vol. 9, no. 45, Dec. 2016, doi: 10.17485/ijst/2016/v9i45/104686.
- [22] G.-C. Zha and C. Paxton, “A Novel Airfoil Circulation Augment Flow Control Method Using Co-Flow Jet,” in 2nd AIAA Flow Control Conference, Portland, Oregon: American Institute of Aeronautics and Astronautics, Jun. 2004. doi: 10.2514/6.2004-2208.
- [23] K. Xu and G. Zha, “System energy benefit using co-flow jet active separation control for a serpentine duct,” *Aerosp. Sci. Technol.*, vol. 128, p. 107746, Sep. 2022, doi: 10.1016/j.ast.2022.107746.
- [24] R. Wang, G. Zhang, P. Ying, and X. Ma, “Effects of Key Parameters on Airfoil Aerodynamics Using Co-Flow Jet Active Flow Control,” *Aerospace*, vol. 9, no. 11, p. 649, Oct. 2022, doi: 10.3390/aerospace9110649.
- [25] J. Julian, Waridho Iskandar, Fitri Wahyuni, Ferdianto, and N. Toding Bunga, “Characterization of the Co-Flow Jet Effect as One of the Flow Control Devices,” *J. Asimetrik J. Ilm. Rekayasa Inov.*, pp. 185–192, Jul. 2022, doi: 10.35814/asiimetrik.v4i1.3437.
- [26] A. Lefebvre, B. Dano, M. Difronzo, W. Bartow, and G. Zha, “Performance of a Co-Flow Jet Airfoil with Variation of Mach Number,” in 51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, Grapevine (Dallas/Ft. Worth Region), Texas: American Institute of Aeronautics and Astronautics, Jan. 2013. doi: 10.2514/6.2013-490.
- [27] B. Dano, D. Kirk, and G. Zha, “Experimental Investigation of Jet Mixing Mechanism of Co-Flow Jet Airfoil,” in 5th Flow Control Conference, Chicago, Illinois: American Institute of Aeronautics and Astronautics, Jun. 2010. doi: 10.2514/6.2010-4421.
- [28] B. Dano, G. Zha, and M. Castillo, “Experimental Study of Co-Flow Jet Airfoil Performance Enhancement Using Discreet Jets,” in 49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, Orlando, Florida: American Institute of Aeronautics and Astronautics, Jan. 2011. doi: 10.2514/6.2011-941.
- [29] J. C. Holt, K. P. Garry, and T. Smith, “Investigation of the Aerodynamic Characteristics of a Lifting Body in Ground Proximity,” in 34th AIAA Applied Aerodynamics Conference, Washington, D.C.: American Institute of Aeronautics and Astronautics, Jun. 2016. doi: 10.2514/6.2016-3881.
- [30] L. Brooke, “Potential Future Developments in Inflated Lifting Body Form LTA Vehicle Hulls,” in 11th AIAA Aviation Technology, Integration, and Operations (ATIO) Conference, Virginia Beach, VA: American Institute of Aeronautics and Astronautics, Sep. 2011. doi: 10.2514/6.2011-6917.
- [31] S. Anoop, R. K. Velamati, and V. R. M. Oruganti, “Aerodynamic characteristics of an aerostat under unsteady wind gust conditions,” *Aerosp. Sci. Technol.*, vol. 113, p. 106684, Jun. 2021, doi: 10.1016/j.ast.2021.106684.
- [32] M. M. and R. S. Pant, “Research and advancements in hybrid airships—A review,” *Prog. Aerosp. Sci.*, vol. 127, p. 100741, Nov. 2021, doi: 10.1016/j.paerosci.2021.100741.
- [33] R. Ravindran et al., “Computational investigation of aerodynamics characteristics over GNVF profile,” presented at the APPLIED PHYSICS OF CONDENSED MATTER (APCOM 2022), Štrbské Pleso, Slovak Republic, 2023, p. 030001. doi: 10.1063/5.0130973.
- [34] M. Atyya, G. M. ElBayoumi, and M. Lotfy, “Optimal shape design of an airship based on geometrical aerodynamic parameters,” *Beni-Suef Univ. J. Basic Appl. Sci.*, vol. 12, no. 1, p. 25, Feb. 2023, doi: 10.1186/s43088-023-00352-1.
- [35] W. Qu, W. Gong, C. Chen, T. Zhang, and Z. He, “Optimization Design and Experimental Verification for the Mixed-Flow Fan of a Stratospheric Airship,” *Aerospace*, vol. 10, no. 2, p. 107, Jan. 2023, doi: 10.3390/aerospace10020107.
- [36] M. M. and R. S. Pant, “Research and advancements in hybrid airships—A review,” *Prog. Aerosp. Sci.*, vol. 127, p. 100741, Nov. 2021, doi: 10.1016/j.paerosci.2021.100741.
- [37] P. Coronado, B. Cuffie, D. Saer, and G.-C. Zha, “Conceptual Design of a Personal Aerial Vehicle Using Co-Flow Jet Airfoil,” in 25th AIAA Applied Aerodynamics Conference, Miami, Florida: American Institute of Aeronautics and Astronautics, Jun. 2007. doi: 10.2514/6.2007-4442.
- [38] A. M. Lefebvre and G. Zha, “Co-Flow Jet Airfoil Trade Study Part I: Energy Consumption and Aerodynamic Efficiency,” in 32nd AIAA Applied Aerodynamics Conference, Atlanta, GA: American Institute of Aeronautics and Astronautics, Jun. 2014. doi: 10.2514/6.2014-2682.
- [39] A. Lefebvre, B. Dano, M. Difronzo, W. Bartow, and G. Zha, “Performance of a Co-Flow Jet Airfoil with Variation of Mach Number,” in 51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, Grapevine (Dallas/Ft. Worth Region), Texas: American Institute of Aeronautics and Astronautics, Jan. 2013. doi: 10.2514/6.2013-490.
- [40] OpenAI, ChatGPT (GPT-5.3), 2026. [Online]. Available: <https://chat.openai.com/>