International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 13 Issue: 06 | Jun 2026 www.irjet.net p-ISSN: 2395-0072

DESIGN AND AERODYNAMIC ANALYSIS OF CIRCULATION CONTROL WING USING COANDA EFFECT FOR LIFT ENHANCEMENT

Solamon M[1], Vinesh K[2], Arjun N P[3], Kiran K murali[4], Jobin John[5]

[123]B.Tech Semester 6, Dept. of Aeronautical Engineering, Toms college of Engineering, Mattakara, Kottayam, Kerala, India
[4]Assistant professor, [5]HOD, Dept. of Aeronautical, Toms college of Engineering, Mattakara, Kottayam, Kerala, India

***

Abstract: - *This project focuses on the design and aerodynamic analysis of a circulation control wing based on the Coanda effect for lift enhancement. The study aims to investigate how the attachment of a high-velocity jet over a curved trailing edge can increase circulation and improve aerodynamic performance. A modified airfoil with a rounded trailing edge and jet slot is modelled and analysed using Computational Fluid Dynamics (CFD). Simulations are carried out under various flow conditions to evaluate key aerodynamic parameters such as lift, drag, and pressure distribution. The results are compared with those of a conventional airfoil to assess performance improvements. The analysis demonstrates that the application of the Coanda effect significantly enhances lift and delays flow separation, especially at higher angles of attack. This study highlights the potential of circulation control wings in improving aircraft efficiency without major structural modifications. It also provides practical insights into airflow behaviour and the effectiveness of advanced aerodynamic control techniques.*

Keywords: - Aerodynamic Analysis, Coanda Effect, Circulation control wing, Computational Fluid Dynamics, Flow separation.

I. INTRODUCTION

Aerodynamic lift plays a crucial role in the performance, efficiency, and stability of aircraft. Conventional airfoils generate lift due to pressure differences created by airflow over their surfaces. However, at low speeds and high angles of attack, flow separation can occur, leading to reduced lift and increased drag. To address these limitations, advanced techniques such as circulation control using the Coanda effect are employed.

The Coanda effect is the tendency of a fluid jet to attach itself to a nearby curved surface and follow its contour. In circulation control wings, a high-velocity jet of air is blown over a rounded trailing edge, increasing circulation and thereby enhancing lift without significant changes in wing geometry. This makes it a promising method for improving aerodynamic performance.

An airfoil is the cross-sectional shape of a wing designed to produce lift when air flows over it. Its geometry plays a vital role in determining aerodynamic characteristics.

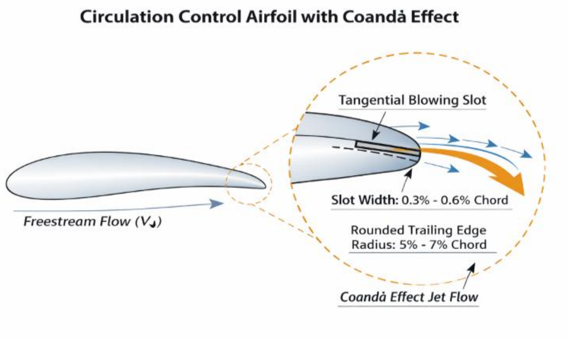


Fig.1.Circulation control airfoil with Coanda effect.

Features of an Airfoil

- Leading Edge: The front portion of the airfoil that first contacts the airflow
- Trailing Edge: The rear end where airflow leaves the airfoil
- Chord Line: An imaginary straight line joining the leading and trailing edges
- Camber: The curvature of the airfoil, which affects lift generation
- Thickness Distribution: The variation of thickness along the chord, influencing strength and airflow
- Upper and Lower Surfaces: Shape difference creates pressure variation for lift.
- Rounded Trailing Edge (Coanda Airfoil): Helps in jet attachment and circulation enhancement.
- Jet Slot (in Coanda Airfoil): Allows high-speed air to flow and adhere to the surface.

Applications of Airfoil

- Aircraft Wings: Used in airplanes to generate lift for flight
- Helicopter Rotor Blades: Provide lift and thrust in rotary-wing aircraft

- Unmanned Aerial Vehicles (UAVs**)**: Improve efficiency and flight control
- Wind Turbine Blades**:** Convert wind energy into mechanical/electrical energy
- Automotive Aerodynamics: Improve vehicle stability and reduce drag
- Marine Applications**:** Used in hydrofoils and propellers for efficient movement in water
- High-Lift Devices**:** Applied in flaps and advanced wing systems for take-off and landing
- Circulation Control Wings: Enhance lift using the Coanda effect in modern aerodynamic systems

This project focuses on the design and CFD analysis of a Coanda airfoil to study airflow behavior and evaluate its effectiveness in improving lift compared to conventional airfoils.

II.METHODOLOGY

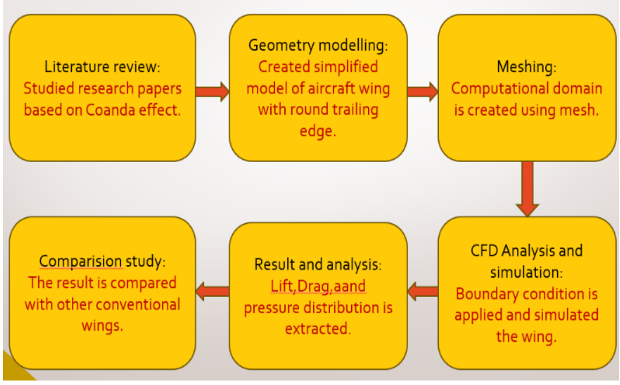


Fig. 2: Flow diagram of methodology

A systematic methodology was adopted to investigate the aerodynamic performance of a conventional airfoil and a Coanda airfoil with circulation control. The study began with an extensive literature review of research papers, journal articles, and technical publications related to the Coanda effect and circulation control wings to gain a thorough understanding of the fundamental aerodynamic principles and previous research findings. Based on the insights obtained from the literature, two airfoil geometries were designed using CATIA software: a conventional airfoil and a modified Coanda airfoil incorporating a rounded trailing edge and a jet slot for circulation control. Following the geometry creation, a computational fluid domain was developed around each airfoil to simulate external airflow conditions accurately while minimizing the influence of boundary effects. The computational domain was then discretized using ANSYS Meshing, where an appropriate mesh was generated to ensure accurate resolution of flow features, particularly in critical regions near the airfoil surface, trailing edge, and jet slot. Subsequently, suitable boundary conditions were applied, including inlet velocity, outlet pressure, wall conditions, and jet velocity at the slot outlet of the Coanda airfoil, to replicate realistic operating conditions. Computational Fluid Dynamics (CFD) simulations were then carried out in ANSYS Fluent to analyse the airflow behavior over both airfoil configurations under various flow conditions. During the simulation process, solver iterations were carefully monitored, and convergence was ensured through the examination of residual plots and iteration histories until the prescribed convergence criteria were satisfied. Upon achieving convergence, important aerodynamic parameters such as lift, drag, pressure distribution, velocity contours, and streamline patterns were extracted from the simulation results. A comprehensive comparative study was then performed to evaluate the aerodynamic performance of the conventional and Coanda airfoils, with particular emphasis on lift enhancement, drag characteristics, and flow behavior. Finally, the results were analysed and interpreted to understand the influence of the Coanda effect on aerodynamic performance, focusing on phenomena such as jet attachment to the curved trailing edge, increased circulation around the airfoil, and delayed flow separation, all of which contribute to improved lift generation and overall aerodynamic efficiency.

III. OBJECTIVES OF STUDY

The primary objective of this study is to develop a comprehensive understanding of the Coanda effect and its application in circulation control airfoils, with particular emphasis on its influence on airflow behavior and aerodynamic performance. The study aims to investigate the fundamental principles governing the Coanda effect, wherein a high-velocity jet flow tends to remain attached to a curved surface, resulting in increased circulation around the airfoil and enhanced lift generation. To achieve this, a modified Coanda airfoil featuring a rounded trailing edge and an integrated jet slot is designed and analysed alongside a conventional airfoil configuration. The project seeks to evaluate how the introduction of a jet flow alters the flow field around the airfoil and influences key aerodynamic characteristics under various operating conditions.

A major objective of the work is to perform detailed Computational Fluid Dynamics (CFD) simulations to visualize and analyse the airflow patterns around both airfoil configurations. Through numerical analysis, the study examines important flow phenomena such as jet attachment to the curved trailing edge, changes in circulation strength, pressure distribution variations, boundary layer development, and the delay or suppression of flow separation. Understanding these flow mechanisms is essential for assessing the effectiveness of circulation control techniques in improving aerodynamic performance. The study also aims to investigate the interaction between the injected jet and the surrounding airflow, and how this interaction contributes to the modification of the pressure field and enhancement of lift.

Another important objective is to quantify and compare the aerodynamic performance of the conventional and Coanda airfoils by evaluating key parameters such as lift coefficient, drag coefficient, lift-to-drag ratio, pressure coefficient distribution, velocity contours, and streamline patterns. By conducting a comparative analysis, the study seeks to identify the advantages and limitations of the Coanda airfoil concept relative to conventional airfoil designs. Particular attention is given to determining the extent of lift augmentation that can be achieved through circulation control and assessing whether the associated aerodynamic benefits outweigh any potential increase in system complexity or energy requirements for jet injection.

Furthermore, the study aims to explore the role of circulation control as an advanced aerodynamic technique for enhancing aircraft performance. The investigation focuses on understanding how increased circulation around the airfoil contributes to greater lift generation without requiring significant changes in airfoil geometry or angle of attack. This aspect of the research is particularly relevant for applications involving short take-off and landing (STOL) aircraft, unmanned aerial vehicles (UAVs), high-lift systems, and other aerospace configurations where improved lift characteristics are desirable. The study also seeks to provide insights into the potential of Coanda-based circulation control systems for reducing flow separation, improving aerodynamic efficiency, and enhancing overall flight performance.

In addition to evaluating aerodynamic performance, the project aims to develop practical knowledge and expertise in the application of CFD tools for aerodynamic analysis. This includes geometry creation, computational domain development, mesh generation, boundary condition specification, numerical simulation, convergence monitoring, and post-processing of results using industry-standard software. Through these activities, the study intends to establish a systematic methodology for analysing circulation control airfoils and generating reliable aerodynamic data.

Ultimately, the objective of this research is to contribute to the understanding of Coanda-effect-based circulation control technology and to assess its feasibility as a means of improving aerodynamic efficiency and lift generation. The findings of this study are expected to provide valuable insights into the aerodynamic behavior of Coanda airfoils and demonstrate their potential for future aerospace applications where enhanced performance, improved controllability, and efficient lift augmentation are required.

IV. GEOMETRY DESCRIPTION

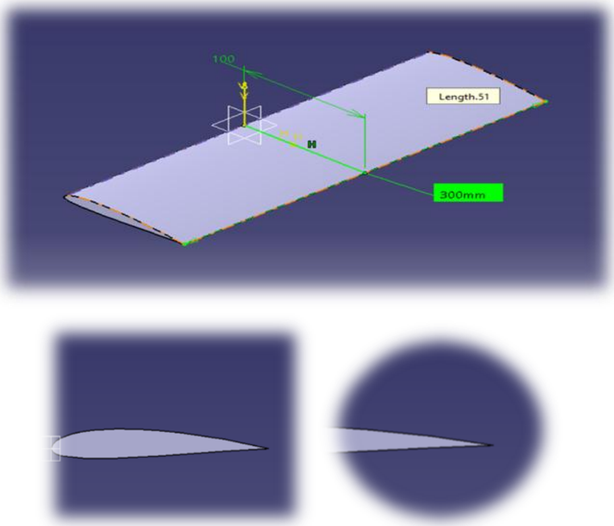


Fig.3: Different geometry of conventional airfoil (NACA 2412)

The airfoil chosen for this design is the NACA 2412 profile, a widely used airfoil known for its balanced aerodynamic performance and suitability for a variety of low- and moderate-speed applications. The designation "2412" provides important geometric information about the airfoil shape. The first digit, 2, indicates that the airfoil has a maximum camber equal to 2% of the chord length. Camber refers to the curvature of the airfoil, and this slight curvature contributes to the generation of lift even at relatively small angles of attack. The second digit, 4, signifies that the location of maximum camber is positioned at 40% of the chord length measured from the leading edge. This placement influences the pressure distribution over the airfoil surface and contributes to stable aerodynamic behavior. The final two digits, 12, indicate that the maximum thickness of the airfoil is 12% of the chord length, providing a good balance between structural strength and aerodynamic efficiency.

For the purpose of this design and analysis, a chord length of 100 mm has been selected, while the span of the wing has been taken as 300 mm. These dimensions provide a practical model size that is suitable for fabrication, testing, and performance evaluation. The selected airfoil incorporates a sharp leading edge, which allows the incoming airflow to be directed smoothly over the upper and lower surfaces of the wing. This geometric feature can influence the flow characteristics around the airfoil and helps establish a clear stagnation point at the front of the profile. The combination of the NACA 2412 geometry, the chosen chord and span dimensions, and the sharp leading-edge configuration makes the airfoil suitable for aerodynamic investigations, offering an effective compromise between lift generation, drag characteristics, and structural practicality.

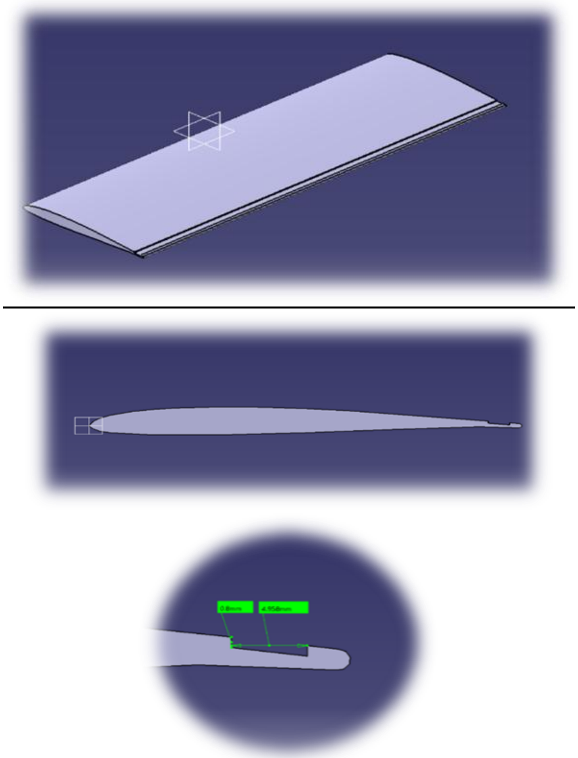


Fig.4:Different geometry of Coanda airfoil(NACA 2412)

The wing configuration is based on the NACA 2412 airfoil, which serves as the reference geometry for the design. A chord length of 100 mm has been selected as the reference length for the airfoil, providing a suitable scale for aerodynamic analysis and experimental evaluation. The overall thickness distribution of the modified airfoil is maintained as close as possible to that of the original NACA 2412 profile in order to preserve its fundamental aerodynamic characteristics, including lift generation, pressure distribution, and structural integrity.

To incorporate flow-control capabilities, the airfoil has been modified near the trailing-edge region. A narrow slot with a height of 0.8 mm has been introduced to act as a jet slot through which airflow can be discharged. The slot is positioned close to the trailing edge so that the injected airflow can effectively influence the boundary layer and wake development in the downstream region. This placement is particularly important because the trailing-edge region plays a significant role in determining the overall lift and drag characteristics of the airfoil. By introducing airflow through the slot, it becomes possible to modify the local flow behavior, delay flow separation, and potentially improve aerodynamic performance.

The trailing edge itself has been designed with a rounded geometry rather than a sharp termination. This rounded trailing-edge configuration provides adequate space for the integration of the jet slot while ensuring smooth airflow guidance toward the exit. In addition, the trailing-edge profile follows a curved contour, allowing the discharged jet to be directed along the surface more effectively. The curved shape promotes a gradual flow turning effect and helps maintain attachment of the jet to the airfoil surface, which can enhance the interaction between the injected airflow and the external flow field. Overall, the modified design combines the proven aerodynamic characteristics of the NACA 2412 airfoil with a rounded and curved trailing-edge configuration featuring a 0.8 mm jet slot, creating a geometry that is well suited for studies involving active flow control, lift enhancement, and drag reduction.

V.FLOW CONDITIONS AND ASSUMPTIONS

The flow analysis is carried out by considering a steady, external aerodynamic flow over the airfoil. The main flow parameters and conditions used in the simulation are listed below:

The computational fluid dynamics (CFD) analysis was carried out under steady-state external flow conditions using air as the working fluid. A free-stream velocity of 60 m/s was specified to represent the incoming airflow over the airfoil. Since the flow velocity is significantly lower than the speed of sound, the flow regime is considered subsonic, with a Mach number less than 0.3. Under these conditions, compressibility effects are negligible, allowing the flow to be treated as incompressible for the numerical simulation. The analysis focuses on understanding the aerodynamic behavior of the airfoil and the influence of the trailing-edge jet on the surrounding flow field.

The computational domain was defined with appropriate boundary conditions to accurately represent the external aerodynamic environment. At the inlet boundary, a uniform velocity inlet condition was applied, ensuring a constant free-stream velocity across the entire inlet plane. At the outlet boundary, a pressure outlet condition corresponding to atmospheric pressure was specified, allowing the airflow to exit the domain naturally without imposing unnecessary constraints on the solution. The operating pressure throughout the simulation was also set to atmospheric pressure, providing a realistic representation of standard ambient conditions.

The airfoil surface was modelled using a no-slip wall boundary condition, which assumes that the fluid velocity relative to the solid surface is zero. This condition is essential for accurately capturing boundary-layer development, wall shear stresses, and flow separation phenomena. In addition to the primary airflow, a high-velocity jet was introduced through the slot located near the trailing edge of the modified Coanda airfoil. The jet flow interacts with the external stream and follows the curved trailing-edge surface due to the Coanda effect, thereby influencing the pressure distribution, enhancing flow attachment, and potentially increasing lift while reducing flow separation.

To account for the effects of turbulence in the flow field, the standard k–ε turbulence model was employed. This model is widely used in aerodynamic simulations because of its robustness, computational efficiency, and ability to provide reliable predictions for a broad range of engineering flow problems. The governing equations were solved using a pressure-based solver, which is particularly suitable for incompressible and low-speed aerodynamic applications. The combination of steady-state assumptions, atmospheric operating conditions, appropriate boundary conditions, and the k–ε turbulence model provides a stable and effective framework for evaluating the aerodynamic performance of the modified Coanda airfoil under subsonic flow conditions.

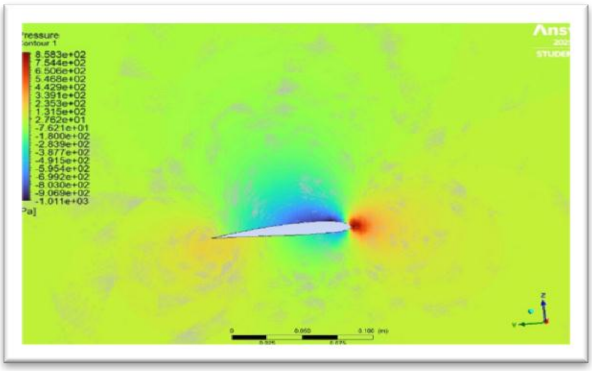


Fig.5: Pressure distribution in normal airfoil

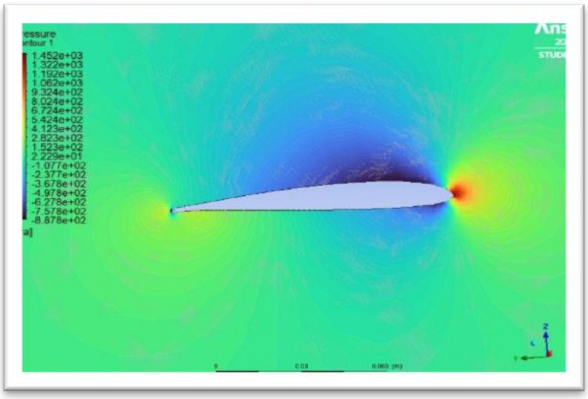


Fig.6: Pressure distribution of Coanda airfoil.

Assumptions:

The analysis is performed under certain simplifying assumptions to make the problem computationally efficient. The flow is assumed to be steady and time-independent. Air is treated as an incompressible fluid with constant properties such as density and viscosity. The study is carried out as a two-dimensional analysis, assuming uniform flow along the spanwise direction. Viscous effects are considered, while body forces like gravity are neglected. The flow is also assumed to be isothermal, meaning no heat transfer takes place. Additionally, the airfoil surface is considered perfectly smooth without any surface roughness. These assumptions help simplify the analysis while maintaining reasonable accuracy in the results.

VI. CFD METHODOLOGY

The aerodynamic analysis of both the conventional airfoil and the Coanda airfoil was carried out through a systematic Computational Fluid Dynamics (CFD) methodology. Initially, detailed two-dimensional geometric models of the conventional airfoil and the Coanda airfoil were developed using CATIA, ensuring accurate representation of their aerodynamic profiles. Following geometry creation, an external computational flow domain was established around each airfoil with sufficiently large upstream, downstream, and far-field boundaries to minimize the influence of boundary conditions on the flow field and to accurately simulate free-stream conditions. The computational domain was then discretized using ANSYS Meshing, where a combination of structured and unstructured meshing techniques was employed to achieve an optimal balance between computational efficiency and solution accuracy. Special attention was given to the airfoil surface, trailing edge region, and areas of expected flow separation by incorporating finer mesh elements to capture detailed flow characteristics. Additional mesh refinement was applied in the boundary layer regions and around the jet slot of the Coanda airfoil, enabling precise resolution of steep velocity gradients and complex flow interactions associated with the Coanda effect. After mesh generation, appropriate boundary conditions were specified for the simulation. A velocity inlet condition of 60 m/s was imposed at the upstream boundary to represent the incoming airflow, while an atmospheric pressure outlet condition was assigned at the downstream boundary.

The airfoil surfaces were modelled as stationary walls with a no-slip boundary condition, ensuring realistic representation of viscous effects. For the Coanda airfoil configuration, a high-velocity jet inlet was defined at the trailing-edge slot to simulate the blowing mechanism responsible for flow attachment and circulation enhancement. The flow solution was obtained using the pressure-based solver available in ANSYS Fluent, which is well suited for incompressible aerodynamic analyses. To accurately model turbulent flow behavior, the standard k-ε turbulence model was selected due to its robustness and widespread use in external aerodynamic simulations. Prior to computation, the solution was initialized using appropriate reference values to provide a stable starting point for the numerical calculations. The solver then performed iterative computations, continuously updating the flow variables until convergence criteria were satisfied. Convergence was monitored through residual reduction and stabilization of key aerodynamic parameters, ensuring the reliability and accuracy of the results. Upon achieving convergence, comprehensive post-processing was conducted to evaluate the aerodynamic performance of both airfoil configurations. Key parameters such as lift force, drag force, lift-to-drag ratio, pressure coefficient distribution, velocity contours, streamlines, and flow-field characteristics were extracted and analysed. The resulting contour plots and streamline visualizations provided valuable insights into the effects of the Coanda jet on flow attachment, circulation enhancement, pressure distribution, and overall aerodynamic efficiency, thereby enabling a detailed comparison between the conventional and Coanda airfoil designs.

VII. ANALYSIS

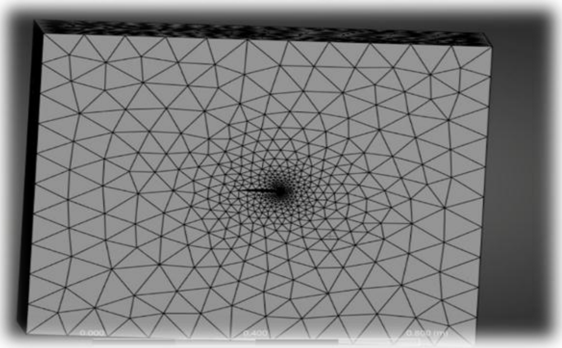
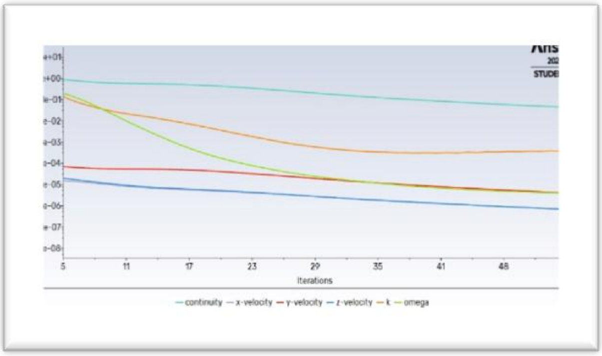


Fig.7: Meshing of normal airfoil.

Graph.1: Iteration curve of normal airfoil.



The iteration curve of the normal airfoil illustrates the convergence behavior of the Computational Fluid Dynamics (CFD) simulation by showing the variation of residual values with increasing iterations. As observed in the graph, the residuals for continuity, velocity components, turbulent kinetic energy (k), and specific dissipation rate (ω) decrease steadily as the solution progresses. At the beginning of the simulation, the residual values are relatively high due to large numerical errors and the initial approximation of the flow field. With each successive iteration, these residuals reduce significantly, indicating that the governing equations are being satisfied more accurately. The x-velocity, y-velocity, and z-velocity residuals decrease to values below 10^{-5}, while the turbulence parameters also show a substantial reduction and stabilization. Although the continuity residual remains comparatively higher than the other variables, it exhibits a consistent downward trend without significant oscillations, indicating acceptable convergence behavior. The smooth decline and stabilization of all residual curves demonstrate that the numerical solution has reached a converged state and that the flow field has become stable. Therefore, the aerodynamic parameters obtained from the simulation, such as lift coefficient, drag coefficient, pressure distribution, velocity contours, and streamline patterns, can be considered accurate and reliable for evaluating the performance of the normal airfoil under the specified operating conditions.

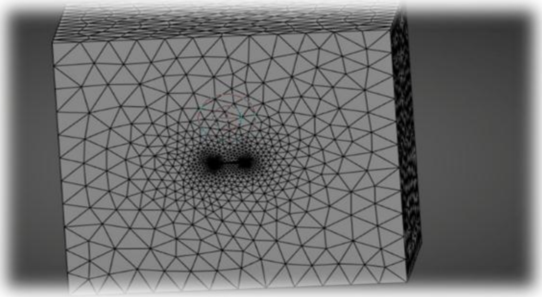
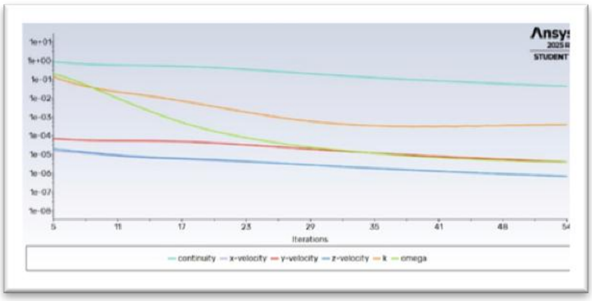


Fig.9: Mesh of coanda airfoil.

Graph.2: Iteration curve of Coanda airfoil.



The iteration curve of the Coanda airfoil illustrates the convergence characteristics of the CFD simulation when a jet flow is introduced at the trailing edge. During the initial stages of the simulation, the residual values are relatively high due to the complex flow physics associated with the interaction between the primary airflow and the high-velocity jet. This interaction creates additional turbulence, flow mixing, and momentum transfer effects, making the numerical solution more challenging to stabilize compared to that of a conventional airfoil. As the iterations continue, the residual values gradually decrease, indicating that the governing equations are being satisfied more accurately and that the solution is moving toward a converged state. In comparison with a normal airfoil, the iteration curve of the Coanda airfoil may exhibit small fluctuations or oscillations throughout the convergence process. These fluctuations arise from the presence of the jet flow, which introduces additional unsteady flow features and increases the complexity of the aerodynamic behavior around the airfoil surface. Nevertheless, the overall downward trend of the residuals demonstrates that the solver is successfully reducing numerical errors and improving solution accuracy with each iteration. Once the residuals reach sufficiently low values and maintain a stable pattern without significant variations, the simulation can be considered converged. This stable convergence indicates that the computed aerodynamic parameters, such as lift coefficient, drag coefficient, pressure distribution, and overall flow behavior around the Coanda airfoil, are dependable and can be confidently used for performance evaluation and further aerodynamic analysis.

VIII.RESULT SUMMARY

Table.1: Result of analysis.

	NORMAL AIRFOIL	COANDA AIRFOIL
LIFT	8.785	9.189
DRAG	1.146	0.921
L/D	7.665	9.977

The CFD analysis of both the conventional airfoil and the Coanda airfoil was successfully carried out to evaluate their aerodynamic performance under identical flow conditions. The results were analysed in terms of iteration convergence, pressure distribution, and lift and drag characteristics.

The iteration curves for both airfoils showed a steady reduction in residuals, indicating proper convergence of the numerical solution. However, the Coanda airfoil exhibited slight fluctuations during the initial stages due to the interaction between the free stream flow and the high-velocity jet at the trailing edge. Despite this, the solution stabilized, confirming reliable simulation results.

The pressure distribution plots revealed a significant difference between the two airfoils. The conventional airfoil showed a typical pressure variation with moderate pressure difference between the upper and lower surfaces. In contrast, the Coanda airfoil demonstrated a much larger pressure difference due to the attachment of the jet flow along the curved trailing edge. This increased suction on the upper surface contributed directly to higher lift generation.

In terms of aerodynamic performance, the Coanda airfoil showed a clear improvement over the normal airfoil. The lift generated by the Coanda airfoil was noticeably higher due to increased circulation caused by the jet flow. Additionally, flow separation was delayed in the Coanda airfoil, allowing it to maintain lift even at higher angles of attack. Although there may be a slight increase in drag due to jet interaction, the overall lift-to-drag ratio was improved, making it more efficient.

The velocity and streamline patterns further confirmed that the jet flow remained attached to the curved surface, validating the presence of the Coanda effect. This attachment enhanced the momentum of the boundary layer and prevented early separation. Overall, the results clearly demonstrate that the implementation of the Coanda effect in a circulation control wing significantly enhances lift performance compared to a conventional airfoil. The study confirms that such modifications can improve aerodynamic efficiency without major structural changes, making it a promising approach for advanced aerodynamic applications.

IX.KEY OBSERVATIONS

- The CFD simulations for both airfoils showed proper convergence, confirming reliable results
- The Coanda airfoil exhibited slight fluctuations in iteration curves due to jet flow interaction
- Pressure difference between upper and lower surfaces is higher in the Coanda airfoil
- Strong suction region observed on the upper surface of the Coanda airfoil
- Jet flow successfully attaches to the curved trailing edge, confirming the Coanda effect
- Flow separation is delayed in the Coanda airfoil compared to the normal airfoil
- Coanda airfoil generates significantly higher lift than the conventional airfoil

- Slight increase in drag may be observed due to jet interaction
- Overall lift-to-drag ratio is improved in the Coanda airfoil
- Enhanced circulation around the airfoil leads to better aerodynamic performance

X.CONCLUSION

This project successfully investigated the design and aerodynamic performance of a circulation control wing using the Coanda effect for lift enhancement. A comparative CFD analysis between a conventional airfoil and a modified Coanda airfoil was carried out under identical flow conditions.

The results clearly demonstrate that the Coanda airfoil provides significant improvement in lift due to increased circulation caused by the high-velocity jet flow. The jet attachment over the rounded trailing edge enhances the pressure difference and delays flow separation, leading to better aerodynamic performance. Although a slight increase in drag is observed, the overall lift-to-drag ratio is improved.

The study confirms that the application of the Coanda effect is an effective method for enhancing lift without major geometric modifications to the airfoil. This makes circulation control wings a promising solution for improving aircraft efficiency, especially in low-speed and high angle of attack conditions.

Overall, the project highlights the importance of advanced aerodynamic techniques and demonstrates the effectiveness of CFD tools in analysing and optimizing airfoil performance.

XI.REFERENCES

1) 1.Airfoil gust load alleviation by circulation control: Yonghong Li, Ning Qin (2020)
2) 2.Analytical and experimental investigation of circulation control by means of a turbulent Coanda jet: E.S. Levinsky, T.T. Yeh (1972)
3) 3.Numerical study of turbulent air and water flows in a nozzle based on a Coanda effect: Youssef El Halal, Cristofer H. Marques, Luiz A.O. Rocha, Liercio A. Isoldi, Rafael de L. lemos, Cristiano Fragassa, Elizaldo D. dos Santos (2019)
4) 4.Numerical simulation and analysis of a Coanda effect circulation control for wind turbine applications: Mohd Faisal Abdul Hamid (2011)
5) 5.Mathematical modelling and numerical investigations on the Coanda effect: A. Dumitrache, F. Frunzulica, T.C. Ionescu (2012)