Numerical Analysis of Aerodynamic Characteristics of Tandem Naca 4412 Airfoils under Heated Wake Influence

Rehaan Pirani, V. Sai Kumar, Ansh Jaiswal
Department of Aeronautical Engineering
Institute of Aeronautical Engineering, Hyderabad,
22951a2135@iare.ac.in, 22951a2152@iare.ac.in, 22951a2158@iare.ac.in

***Abstract* — *This study explores the aerodynamic performance of tandem NACA 4412 airfoils under the influence of a heated wake, focusing on lift and drag characteristics at a Reynolds number of 100000. The investigation combines computational simulations and experimental testing to analyse the impact of heated wake effects, boundary layer separation, and aerodynamic interactions between the airfoils. A three-dimensional computational fluid dynamics (CFD) analysis is performed to investigate the aerodynamic characteristics of a tandem airfoil system under the effect of an unstable wake of a heated cylinder. Both airfoils feature NACA 4412 profiles, and the flow occurs in the ground effect zone with a high lift-to-drag ratio. The study focuses on three control parameters: angle of attack, gap height, and cylinder temperature. The downwash generated by the front airfoil leads to significant aerodynamic interference, affecting the overall performance of the tandem configuration. The primary objective is to enhance understanding of tandem airfoil aerodynamics under isothermal and heated conditions, contributing to the development of next- generation pico-aerial vehicles (PAVs) or unmanned aerial vehicles (UAVs) for thermal surveillance, marine monitoring, and firefighting applications.*

Keywords* — *NACA 4412, Tandem Airfoils, Heated Wake, Flow Separation, Aerodynamics, Computational Fluid Dynamics (CFD), Wind Tunnel Testing

I. INTRODUCTION

A. Background

The aerodynamic behavior of airfoils plays a critical role in determining the efficiency and stability of aerial vehicles. Among various airfoil configurations, tandem airfoil systems have gained significant interest due to their potential to enhance lift generation and improve overall aerodynamic performance. The interaction between multiple airfoils in close proximity influences flow characteristics such as boundary layer separation, wake formation, and lift-to- drag ratio, making tandem configurations a subject of ongoing research.

A key factor affecting tandem airfoils is the presence of wake interference, which can either enhance or degrade aerodynamic efficiency depending on flow conditions. In particular, the introduction of thermal disturbances, such as heated wakes generated by upstream bodies, further complicates the flow dynamics. Understanding how heated wakes influence tandem airfoil performance is essential for applications involving high-temperature environments, such as thermal surveillance, marine monitoring, and firefighting UAVs.

Existing studies have primarily focused on the aerodynamic properties of single airfoils or unheated tandem airfoil systems. However, the impact of temperature- induced wake interactions remain less explored. This study aims to bridge that knowledge gap by conducting a comprehensive investigation into the aerodynamic characteristics of tandem NACA 4412 airfoils under the influence of a heated wake. By utilizing both computational fluid dynamics (CFD) simulations and wind tunnel testing, the research evaluates key aerodynamic parameters, including lift, drag, pressure distribution, and wake behavior, across various angles of attack and gap heights.

The findings from this study will contribute to optimizing airfoil designs for low- Reynolds- number applications, particularly in the development of pico-aerial vehicles (PAVs) and unmanned aerial vehicles (UAVs). By analysing the effects of heated wake interactions, this research will provide valuable insights into improving aerodynamic efficiency, stability, and control strategies for aerial systems operating in thermally dynamic environments

B. Theory

The aerodynamic performance of tandem airfoils is significantly influenced by wake interactions and flow disturbances. The introduction of a heated wake further complicates these interactions by modifying boundary layer behaviour, pressure distribution, and flow stability. This study examines the aerodynamic characteristics of tandem NACA 4412 airfoils under the influence of a heated wake, utilizing both computational and experimental techniques to analyse lift, drag, and wake behaviour.

1. Reynolds Number

The Reynolds number (Re) is a dimensionless number defined in fluid mechanics, which is used to analyse flow patterns under different fluid flow scenarios. It helps to determine if a fluid flow is laminar (smooth and orderly) or turbulent (chaotic and irregular). A low Reynolds number means laminar flow, while a high number means turbulence. Reynolds number is a critical dimensionless number in engineering and fluid dynamics. The formula for Reynolds number is given by: $\boldsymbol{Re = \rho \cdot v \cdot L / \mu}$
where:
ρ = fluid density,
v = flow velocity,
L = characteristic length,
μ = dynamic viscosity

2. Boundary Conditions

The wake generated by the front airfoil in a tandem configuration led to complex boundary layer interactions. When exposed to a heated wake, variations in density and viscosity further alter the flow attachment and separation points. The study considers both isothermal and heated conditions, evaluating their effects on stability, lift augmentation, and drag reduction.

3. Coefficient of Lift

That coefficient of lift, or C_L, is a dimensionless number representing the lift tending to act on an object, such as an airfoil, as it moves through a fluid. It is dependent upon its geometric characteristics, angle of attack, and flow conditions of the fluid. Lift coefficient is very important in aerodynamics for the analysis and design of surfaces giving lift. The equation for the coefficient of lift is as follows: $CL = FL/1/2\rho v2 A$

where:

F_L = lift force,
ρ = fluid density,
v =flow velocity,
A = reference area.

4. Coefficient of Drag

The coefficient of drag is a dimensionless quantity that expresses the resistance of an object moving with some velocity through a fluid, such as air or water. It depends, however, on the shape, surface roughness, and flow conditions around the object. The smaller the drag coefficient, the less drag force to overcome. The drag coefficient aids in the design of efficient vehicles and structures. The formula for the drag coefficient is the coefficient of drag on an object in a fluid flow is:

$Cd = Fd/2\rho v2A$

where:
F_d = drag force,
ρ = fluid density,
v = flow velocity,
A = reference area.

5. Stall Delay and Lift Enhancement

Stall occurs when the airflow separates from the airfoil surface, causing a sharp reduction in lift. In a tandem configuration, the wake from the front airfoil can either delay or accelerate stall on the rear airfoil, depending on flow conditions. The presence of a heated wake may impact boundary layer stability and shift the stall angle.

$\Delta p = p\,upper - p\,lower$

Where:

Δp is the pressure difference between the upper and lower surfaces of the wing, thereby generating lift. A well-constructed corrugated profile thus creates a better pressure difference, improving lift without increasing drag.

6. Three Dimensional Flow Effects

The aerodynamic performance of tandem airfoils is influenced by spanwise flow effects, especially at low Reynolds numbers. Wake vortices, secondary flow structures, and heated disturbances contribute to flow instability. The velocity component in the spanwise direction is given by:

V span=spanwise velocity component

By optimizing tandem airfoil placement and heating conditions, this study aims to control wake- induced instabilities and enhance aerodynamic efficiency for UAV and PAV applications.

II. LITERATURE SURVEY

A. Motivation and Objective

The present study aims to investigate the aerodynamic characteristics of tandem NACA 4412 airfoils under the influence of a heated wake using numerical analysis. The primary aspects to be analysed include lift, drag, pressure distribution, wake interaction, and flow separation to understand how thermal effects alter the aerodynamic performance of the airfoils. The study will provide insights into the modifications in boundary layer behaviour, wake turbulence, and aerodynamic efficiency due to heat-induced variations in airflow properties. The results will contribute to optimizing airfoil configurations for applications in aeronautics where thermal gradients play a significant role in aerodynamic performance.

- To analyse the aerodynamic behaviour of tandem NACA 4412 airfoils under the influence of a heated wake and compare the variations in lift, drag, and pressure distribution with standard, non-heated flow conditions. The study will examine how heat alters the wake characteristics and affects the aerodynamic efficiency of the trailing airfoil.
- To study the wake interactions between the leading and trailing airfoils by analysing flow separation, vortex shedding, and wake turbulence modifications due to the presence of a heated upstream wake. Understanding these effects will help in predicting stability and performance changes in tandem airfoil configurations.
- To assess the influence of temperature gradients on boundary layer development over both airfoils, evaluating how heating affects flow attachment, detachment, and transition characteristics. The goal is to determine whether a heated wake delays or accelerates boundary layer separation, thereby impacting stall behaviour.
- The objective of this paper is to evaluate the effect of heat-induced flow modifications on aerodynamic forces acting on tandem airfoils. This includes studying how temperature influences pressure distribution and lift-to-drag ratio, which are critical factors in determining the performance of tandem airfoil arrangements in practical applications.
- To compare the aerodynamic forces on the leading and trailing airfoils at different angles of attack under heated and non-heated wake conditions. This analysis will identify the key differences in aerodynamic efficiency and help in optimizing tandem airfoil designs for enhanced lift generation and reduced drag in thermally affected environments.

By conducting this numerical study, the findings will offer valuable insights into how heated wake effects influence aerodynamic performance, contributing to the design and optimization of tandem airfoils in various engineering applications such as UAVs, aircraft, and wind energy systems.

B. Literature Review

1. An Application of Active Surface Heating for Augmenting Lift and Reducing Drag of an Airfoil

- Authors: Lucio Maestrello, Forooz F. Badavi, Kevin W. Noonan (1988)
- **Key Findings:** This study explores the effects of active surface heating on an airfoil's aerodynamic performance. The research demonstrates that controlled heating can enhance lift while reducing drag, contributing to improved efficiency in airfoil applications. Also, the results illustrate that the effect of control of the separated flow is very much related to the wave form, the amplitude, and the bandwidth of the temperature function for a given pressure gradient.
- **Reference:** NASA Technical Memorandum (BASA-TM-100563)

2. Experiments on the Forced Wake of Airfoil **Authors:** M. Gharib, K. Williams-Stuber (1988)

- **Key Findings**: The study investigates the wake dynamics of an airfoil under forced oscillations. The effect of initial flow conditions on the wake of an airfoil was examined in this experiment which uses the 'strip heater' technique to externally force the airfoil wake. The strip heaters were used to introduce waves into the top and bottom boundary layers of a thin symmetric airfoil which are subsequently amplified and introduced to the wake. A substantial reduction in drag was achieved as a result of forcing frequencies near the centre of the receptivity range.
- The results indicate significant changes in vortex shedding and wake structures, which influence aerodynamic forces and stability.
- **Reference:** University of California, San Diego, La Jolla, CA 92093, USA ARIAA 88-3840-CP.

3. Lift Characteristics of Two Tandem Airfoils in the Hydrodynamically Self- Excited Wake of a Heated Cylinder Array

•**Authors:** Xiangyu Zhai, Bo Yin, Nader Karimi, Larry K.B. Li, Yu Guan, Ao Wen (2024)

- **Key Findings:** This study examines how a heated upstream cylinder array affects the lift characteristics of tandem airfoils. The findings highlight the impact of thermal wake excitation on aerodynamic forces, emphasizing the role of heat in altering flow behaviour. Also, it was found that heating the cylinder to a temperature above that of the free-stream lead to monotonic decrease in the oscillation frequency and amplitude of Cl for both airfoils. The quantitative lift characteristics were proved to depend strongly on the horizontal spacing between the upstream and downstream cylinders
- **Reference:** J. Mar. Sci. Eng. 2024, 12(5), 832

4. Experimental and Numerical Study on Heatwave Effect Over an Airfoil for Unmanned Aerial Vehicle Applications

- **Authors:** V. Somashekar, Immanuel Selwynraj (2022)

- **Key Findings:** This research explores the influence of heat waves on airfoil performance in UAVs by investigating both numerically and experimentally at the Chord-based Reynolds number and under the influence of various environmental temperatures. The heated wake significantly impacts aerodynamic performance by altering lift and drag coefficients. The study shows that thermal effects can modify boundary layer separation and wake formation, indicating reduced aerodynamic efficiency under heatwave conditions.
- **Reference:** Aircraft Engineering and Aerospace Technology, ISSN: 0002-2667.

5. Numerical Simulation of the Unsteady Wake Behind an Airfoil

- **Authors:** Dean T. Mook, Subhransu Roy, G. Choksi, Bonian Dong (1989)
- **Key Findings:** This work uses numerical simulations to analyse wake interactions behind an airfoil. The study reveals how unsteady wake patterns affect lift and drag, providing insights into thermal wake modifications. It was analysed that in common with previously developed procedures, at each time step a core is added to the wake at the trailing edge, and the cores already in the wake are convicted at the local particle velocity. The innovation of the present method is that, as the cores begin to separate, more cores are added to the system and the circulations around the individual cores are reduced according to a linear interpolation routine.
- **Reference:** ARC Journal, Vol. 26, No. 6, ARC.

6. Thermal Breaking

Authors: Mohammad M. Abdel Karim, Mohamed M. Abdelrahman, Amr Guaily (2023)

•**Key Findings:** This study introduces a novel approach to aerodynamic braking using thermal effects, which involves heating a portion of the air foil's upper surface to achieve the braking effect. The research suggests that heating the surface of an airfoil can create drag modulation and decreases the lift coefficient enhancing the braking effect on the airfoil. This thermal air braking concept effectively alters aerodynamic performance by modifying airflow characteristics around the airfoil which could be useful in-flight control applications.

•**Reference:** NILES, 10.1109/NILES59815.2023.10296779.

7. Numerical Simulations of Ultra-Low-Re Flow Around Two Tandem Airfoils in Ground Effect: Isothermal and Heated Conditions

- **Authors:** Bo Yin, Yu Guan, Ao Wen, Nader Karimi, Mohammad Hossein Doranehgard (2021)
- **Key Findings:** The study numerically investigates the aerodynamic performance of tandem airfoils operating in ground effect under heated and isothermal conditions. Here, the numerical scheme was firstly validated with the experimental data. A parametric study with different heating temperatures and heating areas was carried out and it was found that the lift and drag coefficients both drop with surface heating, especially at a larger angle of attack. Results indicate that thermal influences alter flow separation and aerodynamic efficiency.
- **Reference:** Journal of Thermal Analysis and Calorimetry Volume 145, pages 2063– 2079, (2021), https://doi.org/10.1007/s10973-020-09987-z

8. Numerical Investigation of the Aerofoil Aerodynamics with Surface Heating for Anti- Icing

- **Authors:** Bowen Li, Qiangqiang Sun, Dandan Xiao, Wenqiang Zhang (2022).
- **Key Findings:** The study numerically simulates the two-dimensional flow field around two tandem NACA 0012 airfoils in ground effect, at a Reynolds number low enough to be relevant to pico-aerial vehicles (PAVs). Heating the fore airfoil decreases the lift coefficient without significantly affecting the drag coefficient, resulting in a reduced lift-to-drag ratio. The heated wake alters the pressure distribution, impacting the overall aerodynamic performance of the tandem airfoil system.

III. METHODOLOGY

A. Design Concept

The study investigates the aerodynamic characteristics of tandem NACA 4412 airfoils under the influence of a heated wake. The design concept involves placing a heat generator upstream to mimic thermal wake effects and analysing its impact on aerodynamic forces. The study integrates both experimental and numerical approaches, with physical wind tunnel testing and force and moment measurement using a 6-component wind tunnel balance and Computational Fluid Dynamics (CFD) simulations to evaluate performance changes in lift, drag, wake interactions, and boundary layer behaviour.

B. Geometry and Parameters

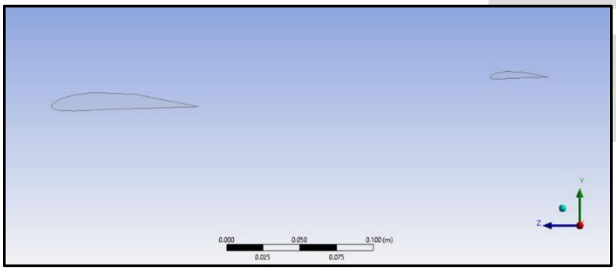
The design process begins with the development of the NACA 4412 airfoil, ensuring optimal chord length for aerodynamic efficiency and span for the 3D geometry. The chord length and span for both the airfoils were finalized after studying different real-world aircrafts, especially the Airbus A320 as in the ratio of the chord lengths for the wing and horizontal tail, surface areas, tail volume ratio, the distance between the two airfoils in terms of chord length and so on; and were modelled based on the wind tunnel constraints. The geometry is constructed as follows:

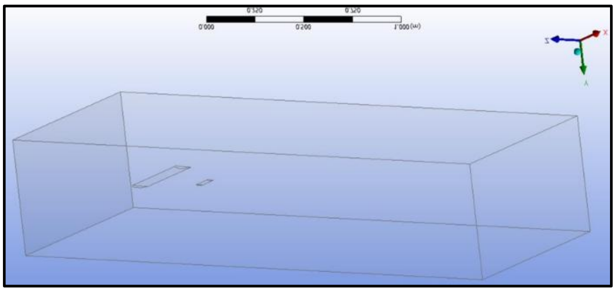
- Airfoil Design: The NACA 4412 profile is generated using the Airfoil Coordinate Data with a chord of 10cm for the front airfoil (wing) and a chord of 4cm for second airfoil (horizontal tail) and is imported into ANSYS Workbench for modelling. The airfoil shape is sketched on the YZ-plane in the Design Modeler of Workbench, ensuring precise representation of the aerodynamic contour.

- Wing Geometry: The airfoil is extruded along the spanwise direction (in X-direction) to define the 3D wing model. The span, taper ratio, and other geometric parameters are adjusted according to the study requirements for tandem airfoil configurations and a span of 40cm is defined for wing and 10cm for tail.

- Heat Generator Placement: A heat source is designed and positioned upstream of the airfoil to induce a controlled heated wake. The heat generator is modeled to create a localized temperature gradient, simulating real-world thermal disturbances. This is done through the placement of Nichrome wire near the leading edge of the wing which is heated by supplying current which heats up the wire. The temperature of the wire is controlled by the amount of current flowing through it which in turn controlled by a device called ‘Temperature Controller’. In this device, we can input the desired wire temperature based on which it supplies the required current through the wire.

- Tandem Airfoil Configuration: The two NACA 4412 airfoils are placed in tandem with a gap of 27cm between them which was considered based on the study of different real- world aircrafts, especially the Airbus A320. The trailing airfoil experiences the heated wake generated by the upstream heat source, allowing for a comparative aerodynamic study.





C. Meshing

To ensure accurate numerical simulation, a high-quality computational mesh is generated for the CFD analysis of the tandem NACA 4412 airfoils under heated wake conditions.

1. Mesh Preparation:

The airfoil geometry, along with the heat generator, is exported into ANSYS Fluent for meshing and simulation.
A computational domain is created to represent the airflow region, ensuring that boundary effects do not interfere with aerodynamic results.
The domain extends sufficiently to capture wake formation and vortex interactions and is of the dimensions 2.5x0.6x0.6m (0.5m in -X direction, 2.5m in +X direction and 0.3m in all other directions).

2. Meshing Method:

An unstructured mesh is applied on the airfoil surface to accurately capture the flow and other important effects by ensuring high resolution mesh around flow separation and wake regions. A mesh sizing of 0.01m is finalized for face sizing for the airfoils after the grid independence study.

An unstructured mesh is used in the far-field to optimize computational efficiency while maintaining aerodynamic accuracy. The element size and the maximum element size were finalized at 0.05m and the growth rate was fixed at 1.15 after the results from the grid independence study.

Refinement zones are implemented at critical locations, including the leading edge, trailing edge, and wake region, as well as around the heat generator, to resolve thermal flow disturbances effectively. The mesh size was observed as, number of cells – 1162516, Faces - 2344780, Nodes – 204448.

3. Mesh Quality Tests:

the mesh undergoes skewness, aspect ratio, and element quality checks to prevent errors in numerical computation and a minimum orthogonal quality of 0.208 was observed in less than 1% of cells and an average aspect ratio of 14.84 was maintained.

Grid independence tests are conducted to ensure that the results remain consistent with increasing mesh refinement and are conducted as follows:
First the 3D wing was studied and max. element size of 0.2m and face sizing of 0.05m was tested and the solution did not converge for over 1000 iterations. The lift force was obtained as 1.589N and drag force as 0.158N.
Now, a mesh with max. element size of 0.1m and face sizing of 0.03m was tested and the solution converged after 209 iterations. The lift force was obtained as 1.604N and drag force as 0.153N.
Again, a mesh with max. element size of 0.05m and face sizing of 0.01m was tested and the solution converged after 182 iterations. The lift force was obtained as 1.603N and drag force as 0.151N.

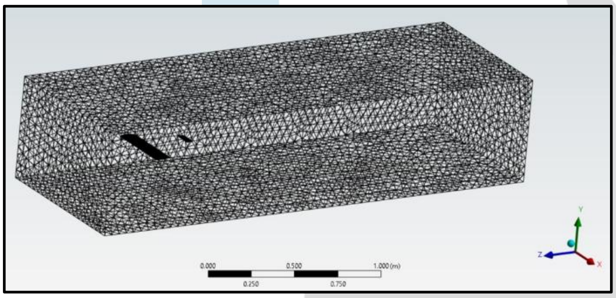
Hence the above mesh was used for further calculations as we can clearly observe that there was less than 1% change in results with mesh enhancement and therefore it was understood that no further mesh refinement is necessary nor is suggested based on the cost of computation with further mesh refinement.

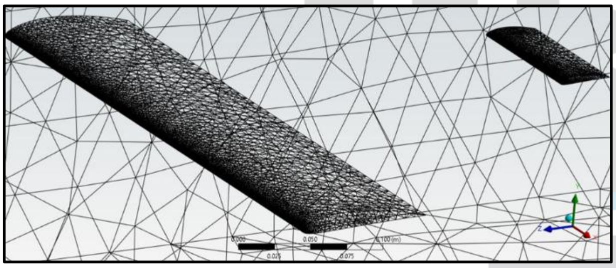
Similarly, the 3D tail was studied with the above mesh characteristics and the lift force was obtained as 0.125N and drag force as 0.0195N and the solution converged after 180 iterations.

4. **Wind Tunnel Testing:**

For the wind tunnel testing, the 3D airfoils are constructed by 3D printing technology where PLA material is used for fabrication due to its unique material properties and ease in handling and affordability and availability.

The forces and moments are measured using a 6-component wind tunnel balance by Sunshine Measurements Pvt. Ltd. which gives us results such as Lift, Drag, Side force and pitching, yawing and rolling moments using strain gauges and load cells.





IV. SOLVER SETUP AND TESTING

For CFD simulation, the solver was designed as follows:
Application: Ansys Fluent
Setting: 3d, double precision, pressure-based, SST k-omega
Heat transfer: Enabled
Turbulent Specification Method: Intensity and Viscosity Ratio
Turbulent Intensity: 5%
Turbulent Viscosity Ratio: 10
Equations: Flow, Turbulence and Energy.
Numeric: Absolute velocity formulation Pressure-velocity coupling: Coupled.
Discretization Scheme: Pressure – Second order, Momentum, Turbulent Kinetic Energy, Specific Dissipation Rate, Energy - Second Order Upwind.

The Reynold number was fixed at 100000 and hence, based on the density and viscosity variation with temperature, the velocity of flow was calibrated so as to have the Reynolds number fixed at 100000. This can be observed from the table below:

Temperature	Dynamic Viscosity	Density	Calibrated Velocity
20	18.13x10^-6	1.204	15.15
25	18.37x10^-6	1.184	15.53
30	18.6x10^-6	1.164	16
40	19.07x10^-6	1.127	16.92
50	19.53x10^-6	1.093	17.86
60	19.99x10^-6	1.06	18.86

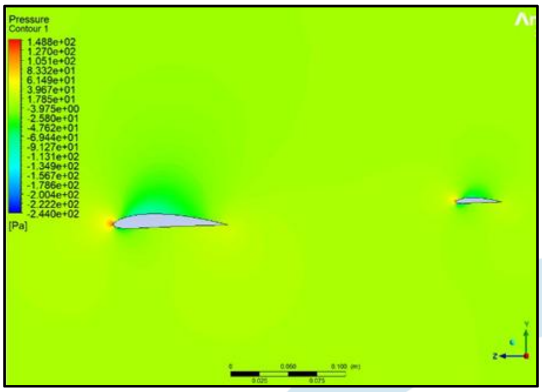
For the wind tunnel testing, the tandem wings were fixed in the wind tunnel as below and the wind tunnel balance was attached to them as below;

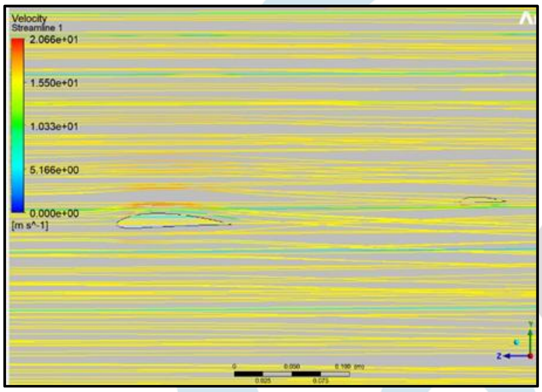
V. EXPECTED RESULS AND DISCUSSION

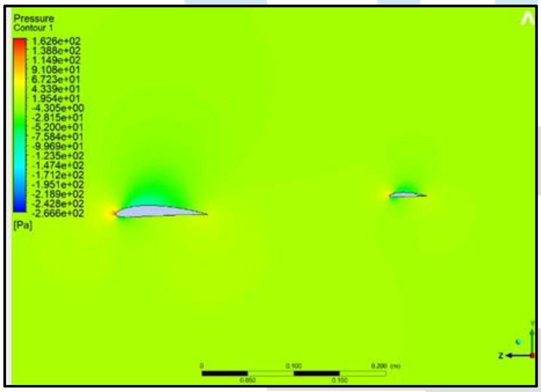
This study examines the aerodynamic characteristics of tandem NACA 4412 airfoils under the influence of a heated wake, focusing on lift, drag, pressure distribution, and wake interaction. The computational and experimental analysis suggests that the introduction of a heated wake significantly alters the aerodynamic efficiency of the tandem airfoil system. The findings reveal that temperature gradients influence boundary layer behaviour, impacting flow stability, vortex formation, and stall characteristics.

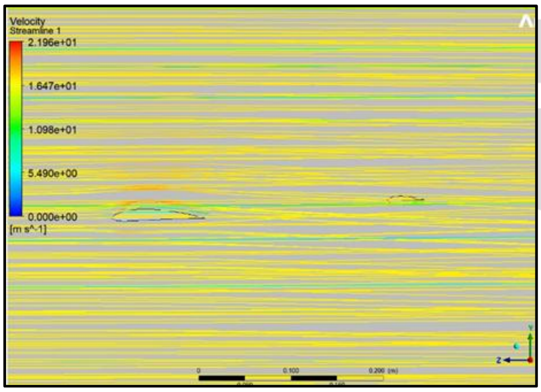
A. Wake Influence on Aerodynamic Performance

The heated wake generated by the upstream airfoil alters the flow characteristics of the trailing airfoil. The downwash effect increases lift in certain conditions, but excessive heating leads to higher drag and turbulent wake structures. CFD simulations indicate that the aerodynamic performance of the trailing airfoil is highly dependent on the intensity of the heated wake and its interaction with the boundary layer.









B. Lift Enhancement and Stall Delay

The results show that, at moderate heating levels, the wake energizes the trailing air foil's boundary layer, delaying flow separation and increasing lift production. Compared to conventional tandem airfoil configurations, the heated wake helps sustain lift at higher angles of attack before reaching stall conditions. However, excessive heating can destabilize the airflow, accelerating stall on the trailing airfoil due to increased wake turbulence.

C. CFD Validation and Experimental Correlation

Preliminary ANSYS CFD simulations closely align with experimental wind tunnel data, validating the computational model. Smoke visualization and dye-based flow visualization techniques confirm the wake interactions predicted in the simulations, reinforcing the impact of heated wakes on aerodynamic performance. The results from the CFD simulation are tabulated below;

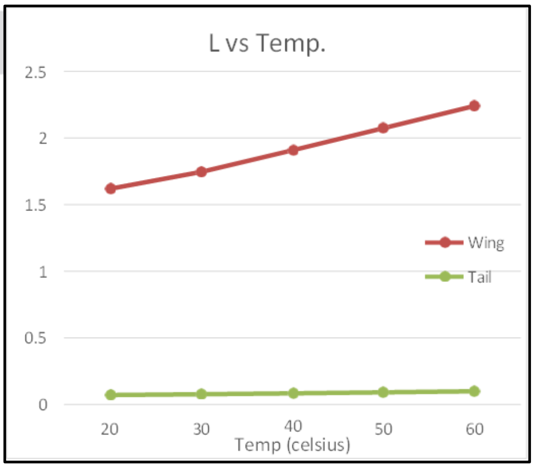
Temperature (°C)	Wing Lift (N)	Tail Lift (N)	Wing Drag (N)	Tail Drag (N)
20	1.6203	0.0713	0.151	0.0202
30	1.747	0.077	0.163	0.0218
40	1.909	0.084	0.179	0.0238
50	2.076	0.0914	0.194	0.0259
60	2.243	0.0988	0.21	0.028

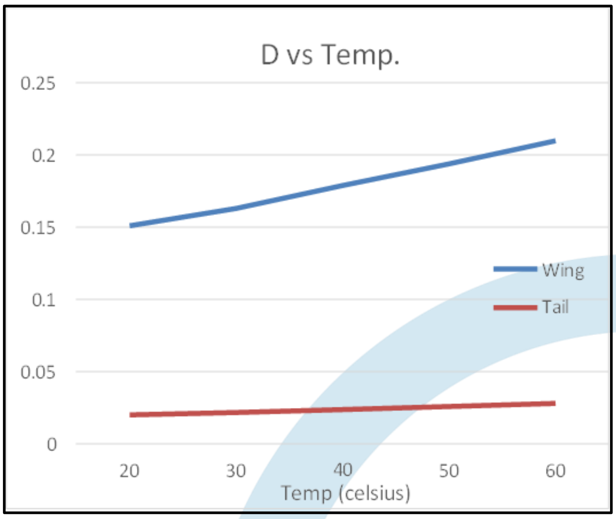
Similarly, the results from the wind tunnel testing are also tabulated below;

Temperature (°C)	Lift (N)	Drag (N)
30	1.68	0.158
40	1.83	0.173
50	1.96	0.188

Hence, from the above two tables, we can observe that there is about 3% difference between the wind tunnel testing results and the CFD simulation results. Therefore, the study is validated both from the CFD data and the experimental data. Also, the results for change in lift and drag with variation in temperature can be graphically represented as below;

Hence, from the two given graphs, we can clearly see the increase in Lift force with increase in temperature as expected based on theory and as stated earlier and also a slight increase in drag with increase in temperature. Therefore, the findings from computational and experimental analysis suggests that the introduction of a heated wake alters the aerodynamic efficiency of the tandem airfoil system.





VI. REFERENCES

1. Lucio Maestrello, Forooz F. Badavi, Kevin W. Noonan, An application of active surface heating for augmenting lift and reducing drag of an airfoil. (BASA-TM- 100563) (1988).
2. M. GHARIB AND K. WILLIAMS-STUBER, Experiments on the forced wake of an airfoil, San Diego, La Jolla, CA 92093, USA ARIAA 88-3840-CP (1988).
3. Xiangyu Zhai, Bo Yin, Nader Karimi, Larry K.B. Li, Yu Guan1, AO Wen, Lift characteristics of two tandem airfoils in the hydrodynamically self-excited wake of a heated cylinder array, J. Mar. Sci. Eng. 2024, 12(5), 832 (2008).
4. V. Somashekar, Immanuel Selwynraj, Experimental and numerical study on heatwave effect over an airfoil for unmanned aerial vehicle applications, Aircraft Engineering and Aerospace Technology, ISSN: 0002-2667 (2022).
5. Dean T. Mook, Subhransu Roy, G. Choksi, Bonian Dong, Numerical Simulation of the Unsteady Wake Behind an Airfoil, Vol. 26, No. 6, ARC (1989).
6. Mohammad M. Abdel Karim, Mohamed M. Abdelrahman, Amr Guaily, Thermal Air Braking, 10.1109/NILES59815.2023.10296779 5th Novel Intelligent and Leading Emerging Sciences Conference (NILES) (2023).
7. Bo Yin, Yu Guan, Ao Wen, Nader Karimi, Mohammad Hossein Doranehgard, Numerical simulations of ultra-low-Re flow around two tandem airfoils in ground effect: isothermal and heated conditions, Journal of Thermal Analysis and Calorimetry Volume 145, pages 2063– 2079, https://doi.org/10.1007/s10973-020-09987-z (2021).
8. Bowen Li, Qiangqiang Sun, Dandan Xiao, Wenqiang Zhang, Numerical Investigation of the Aerofoil Aerodynamics with Surface Heating for Anti-Icing, Aerospace 2022, 9(7), 338.