IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 5, Issue 3, May 2025
Impact Factor: 7.67

CFD Simulation of Spray Mechanism for De-Iceing on Wings Leading Edge

Pratiksha Itankar, Prachi Wankhede, Sanika Chandanshiv, Juhi Bagde
Aeronautical Engineering Department
Priyadarshini College of Engineering, Nagpur, India

Abstract: *This report summarizes the computational fluid dynamics research conducted towards airflow circulation estimation and forecast around an aerodynamic surface. The subject matter of this study is the NACA 641-212 airfoil. A model of the wing with the original geometrical ratios was integrated to improve exposition clarity. The simulations were done in ANSYS Fluent and involved the testing of the airfoil under two scenarios: a single-phase flow scenario with only air and the other scenario of air plus water droplets as a two-phase flow. To prove the model, we applied the Spalart-Allmaras turbulence model for an extreme airfoil surface, performed water droplet dispersion with air within the Distinct Phase Model (DPM) framework, which is standard in such verifications. We conducted a three-dimensional validation test of the NACA 641-212 airfoil using ANSYS Fluent. We simulated the three-dimensional airfoil, and recorded more representative flow behaviour especially around the edges and surfaces. We utilized cutting-edge (Ansys Fluent) simulation tools to test how rain behaves when coming into contact with a 3D wing, and compared our results to real test data to validate that our model was accurate.*

Keywords: CFD; Spalart–Allmaras; discrete phase model (DPM); structured mesh; two-phase flow; Ansys Fluent

I. INTRODUCTION

Today, Computational Fluid Dynamics (CFD) plays a vital role in fluid mechanics. It's become a go-to tool for analysing flow behaviour across a wide range of applications. In recent decades, one particular area of focus has been the impact of rain on aerodynamic surfaces. In recent years, researchers have taken a closer look at how rain affects the performance of aerodynamic surfaces, like those on aircraft. When raindrops enter the airflow around something like a wing, they act as a second phase mixing with the air and changing how it flows. This seemingly simple interaction can actually have a big impact: it disrupts the smooth airflow, which reduces the wing's efficiency and overall aerodynamic performance. That's a serious concern, especially when it comes to flight safety and fuel efficiency.

In this study, we're looking at how air alone (one-phase flow) and a mix of air and water (two-phase flow) behave around a specially designed wing shape. We're keeping the Reynolds number the same with respective to base paper so we can clearly see the differences in aerodynamic performance. To analyse this, we're using computer simulations. For the simulations, we've chosen the Spalart–Allmaras turbulence model, which helps us accurately understand how the airflow behaves around the wing.

In this study, we used a wing based on the NACA 641-212 airfoil, which belongs to the six-digit airfoil family designed to increase laminar airflow over the wing. First, we took the 2D shape of the airfoil and turned it into a 3D wing by giving it a semi-span of 0.73 meters and a chord length of 0.43 meters, to make testing easier.

The study began by running computer simulations to analyse how air (in a single-phase flow) and a mix of air and water (in a two-phase flow) move around two wings. Importantly, they kept the Reynolds number the same for both wing models to ensure a fair comparison.

Once the simulations were complete, they built a physical wing model, again using both air-only and air–water flows, while carefully keeping the Reynolds number consistent with the simulation setup.

The next step was to compare our model's results in two parts and see how they matched with the results from the reference (base) paper. We did this for both air flow alone (one-phase) and air mixed with water droplets (two-phase). The main goal was to check how close our results were to those in the base paper. This helps us understand if our simulation is good enough to trust, which can save time and effort. It also shows if there are any differences between our model and the one used in the base paper.

The final step was to compare the results of one-phase flow (just air) with two-phase flow (air mixed with water droplets). This comparison was done for all the simulation cases and the reference data to understand how the presence of water droplets affects the aerodynamic performance of the wing.

II. NUMERICAL METHODS AND SIMULATION

Fluid flow, like the movement of air or water, is described using mathematical equations. The most complete and widely used equations for this are the Navier–Stokes (N–S) equations, which work under the idea that fluids are continuous and smooth. However, these equations are very complex and can't be solved exactly in most real situations. So, we use computer-based methods to solve them approximately by breaking the problem into smaller parts — this process is called discretization.

To make things more manageable, especially when dealing with turbulent flow (which is chaotic and unpredictable), we simplify the equations. In this study, we used a method called the Reynolds-Averaged Navier–Stokes (RANS) equations. This method separates the flow into two parts: one that shows the average or steady behaviour over time, and another that captures the small, quick fluctuations (turbulence).

Since it's not practical to model every tiny detail of turbulence, we use turbulence models to estimate its effects. In our case, we used eddy viscosity models, which come in two types: one-equation and two-equation models, which help estimate how turbulence affects the flow without having to simulate every detail.

The Spalart–Allmaras turbulence model is a type of one-equation model used to predict turbulence in fluids, especially for aerospace applications. It's designed to be efficient for flows near surfaces, like the ones that happen over wings or other parts of an aircraft. Unlike other turbulence models, the Spalart–Allmaras model makes it simpler and faster to compute. The model is particularly useful for situations where the flow is attached (not separated) and where the flow starts to separate slowly.

To calculate how multiple (multiphase flow) type of materials (like air and water droplets) move together in a flow, we use a method called the Eulerian-Lagrangian approach. In this method, the main fluid (like air) is treated as a continuous substance, and we solve special equations (the time-averaged Navier–Stokes equations) to understand how it moves. The droplets or particles (like water drops) are tracked one by one as they move through the air. These tiny particles don't just float around — they can interact with the air, changing its speed, temperature, or even mixing with it. So basically, we calculate how the air flows everywhere, and at the same time, we follow each droplet through that flow to see how both the air and droplets affect each other.

This method is called the Discrete Phase Model (DPM). It's used to simulate how small particles, like water droplets or bubbles, move through a fluid such as air. These particles are usually treated as tiny balls floating in the air. We track the path of each droplet, and also see how it gains or loses heat or mass — like when a droplet warms up, cools down, or starts to evaporate. The simulation can also include how the droplets and the air interact with each other. So, not only does the air affect how the droplets move, but the droplets can also change the way the air flows. To know where each droplet will go, we use a simple idea: the forces on the droplet decide how it moves, just like how gravity or air resistance affect the motion of a falling ball. This is done in what's called a Lagrangian frame, where we follow each droplet along its path.

In our study, we ran both one-phase (just air) and two-phase (air with water droplets) simulations for both the full-size wing models. We used the same airflow speed for all tests, matching a Reynolds number of 2.1×10^5, which was also used in real wind tunnel experiments. We compared results like lift, drag, pressure, and aerodynamic efficiency to see how well the wing performed.

As mentioned earlier, the wing was designed using an NACA 641-212 airfoil shape. The wing's size is given in terms of the mean aerodynamic chord (MAC), which is a standard way to describe the average width of a wing. For the

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429 Volume 5, Issue 3, May 2025 Impact Factor: 7.67

original full-size version of the wing, the MAC was 0.43 meters. This study used ANSYS Fluent, a commercial CFD (Computational Fluid Dynamics) software, to carry out the simulations.

A 3D simulation domain was created, in that 3D simulation domain we did a geometry of wing in (Ansys fluent) design modeler and generate a 3D meshed of airfoil. The RANS equations (Reynolds-Averaged Navier-Stokes) were solved to analyse airflow around the wing. The domain's size was defined using the mean aerodynamic chord (MAC) as a reference unit, and its dimensions were carefully matched to the size of the wind tunnel test section where the physical model of the wing was tested. This ensured consistency between the simulation and the real-world experiment. The surfaces of the simulation space were rectangular-shaped, the side surface of rectangle which is vertical surface is about 6.5MAC but we take in metres so it will be in 6.5m and the horizontal surface is 20m. The wing is placed little closer to rectangle inlet so the distance between in rectangle inlet from wing is 7m which is horizontal distance and the vertical distance of wing from upper and lower surfaces of rectangle both the distance is 3.25m. We take a NACA641212 six-digit airfoil to made a wing so we extruded that airfoil up to 1.7m.

6.5MAC

20MAC

Figure 1.a side surfaces of rectangle (computational domain)

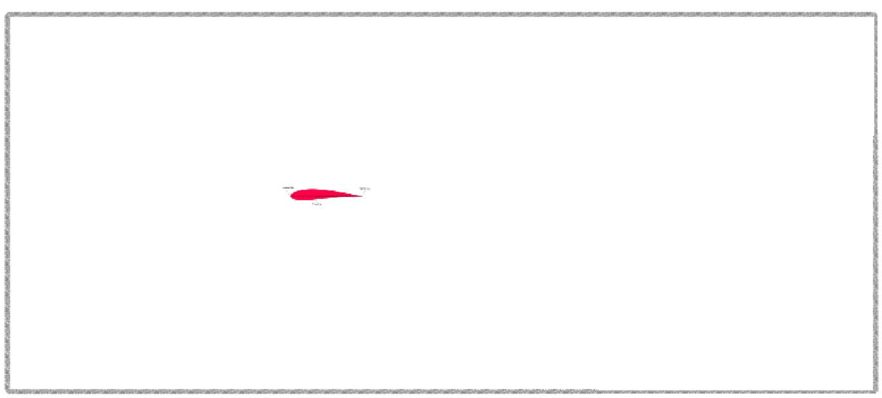

Figure 1.b Wing placed inside rectangle (computational domain)

To get accurate results from the simulation, the shape and size of the airflow area had to match the real setup used in the wind tunnel. That's why an "H-type" layout was chosen for the simulation model. The mesh (which is like a grid used to break the area into small pieces for calculation) became more detailed as it got closer to the wing's surface. The very first layer of the mesh, right next to the wing, was made extremely thin—just 1.96×10^{-5} meters. This was done to capture the thin layer of air that clings to the wing (called the boundary layer). In the simulation, the area where air flows around the wing was divided into tiny 3D boxes called hexahedra. This made a clean and organized grid, or mesh, that helps the computer calculate how air moves. To figure out how many mesh cells were enough for good

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429 Volume 5, Issue 3, May 2025 Impact Factor: 7.67

results without wasting computer power, a mesh sensitivity test was done. the test was done with the wing kept flat (zero angle of attack). The results showed: The full-size wing model worked well with 4.8 million mesh cells.

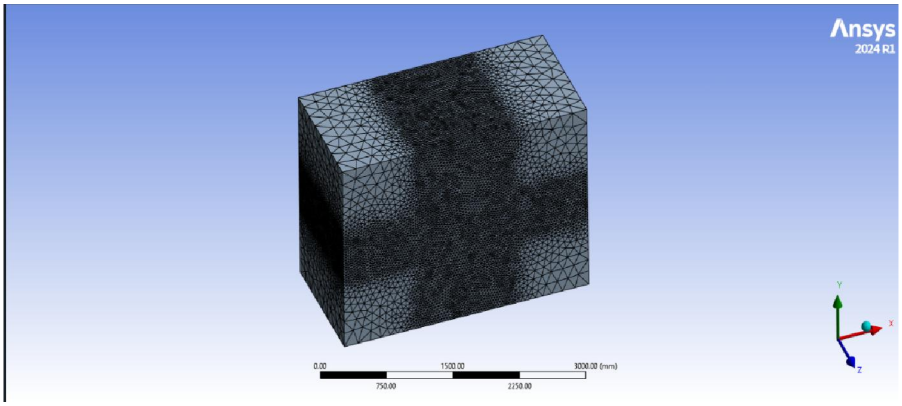

Figure 2. (a) 3D Meshed

After creating the mesh, the next step was to set up the simulation. This included choosing the turbulence model, setting the boundary conditions, and selecting the numerical methods the software would use. As mentioned earlier, the Spalart–Allmaras model was used for all the simulations. This model is a good choice for aerospace problems because it works well for flows that stay close to surfaces (like over a wing) and for cases where the air separates slowly. It's especially useful when the flow stays mostly smooth and attached to the surface.

For the airflow simulation, the pressure-based solver in Ansys Fluent was used. Since the air movement didn't change over time, the flow was treated as steady (time-independent).To accurately link how pressure and velocity interact, a coupled pressure–velocity method was applied. To reduce errors in the simulation, a second-order upwind scheme was used right from the beginning. This method gives more accurate results, which was important for comparing with real life testing of wind tunnel experiment. The air properties were set as for a Temperature: 284.18 K, Density: 1.1561 kg/m³, Viscosity: 1.7701×10^{-5} kg/(ms) and the airflow speed at the inlet was 7 m/s for the original (full-size) model. To simulate the movement of water droplets in the air, the Discrete Phase Model (DPM) was used. This model helps track how droplets behave in the airflow. To make the simulation more realistic, wall-film conditions were applied to the wing surface. In the simulation, the water droplets (the second phase) were treated as unsteady and a two-way turbulence interaction was used.

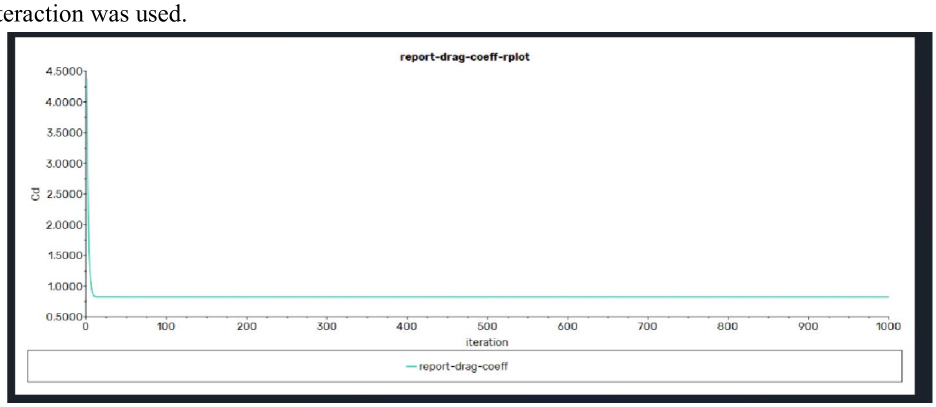

Figure 2. (b) Drag coefficient

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 5, Issue 3, May 2025
Impact Factor: 7.67

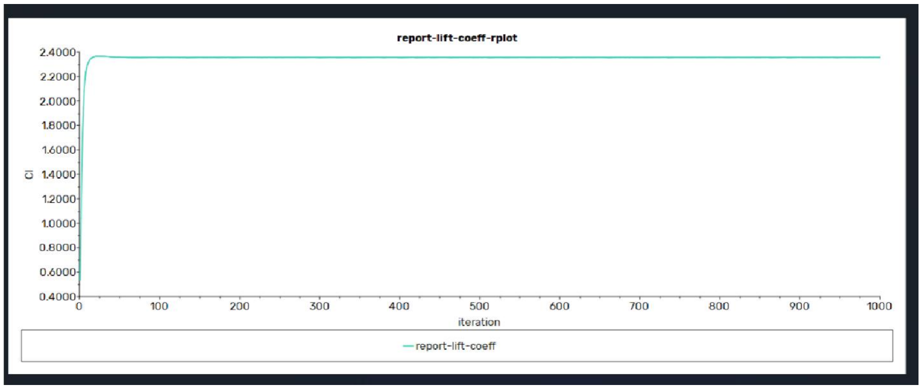

Figure 2. (c) Lift coefficient

In the simulation, water droplets were sprayed into the airflow through a surface placed at the top half of the inlet, and the spray direction was perpendicular to the airflow. The droplets were released at regular time intervals of 0.001 seconds. Each droplet had a size of 0.0005 meters in diameter. The speed of the droplets moving with the airflow was set to 7 m/s for the original model.

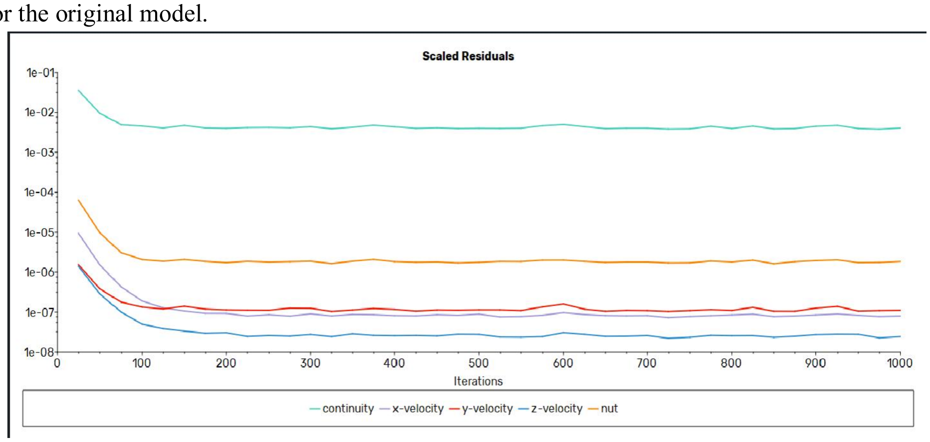

Figure 2. (d) Scaled Residuals

III. RESULT

We used the Spalart–Allmaras turbulence model to run computer simulations. To make sure the results were reliable, compared the simulation data with results from real life wind tunnel tests and reference data. Next, we compared two types of simulations: One where only air was flowing (one-phase flow), and one where both air and another substance (like water droplets) were flowing together (two-phase flow). The study had main goals is to understand how the added particles affected the airflow and how the wing performed aerodynamically. In the part of the study, computer simulations were carried out on the original full-size wing and the main results focused on the wing's drag and lift coefficients—basically, how much air resistance the wing faced and how much upward force it generated.

When looking at the results, we can see that at the same Reynolds number, the lift values (lift coefficient) for the real-life wind tunnel experiment data and reference data were very close to those of the original full-size wing, just slightly lower. The results focused on how the lift and drag coefficients changed. Overall, the lift and drag coefficient in the two-phase flow behaved slightly similar like a one-phase.

The results clearly show that the presence of water droplets (the second phase) reduced the wing's aerodynamic performance. The drop in lift-to-drag ratio (L/D), which indicates a decline in aerodynamic efficiency, was observed in real life wind tunnel testing this decrease is mainly due to the presence of a second phase. The wing experienced around a 4% reduction in aerodynamic efficiency due to this second phase.

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 5, Issue 3, May 2025
Impact Factor: 7.67

When air flows quickly over the top of the wing, it usually creates a low-pressure area that helps generate lift. But when water droplets are present (in the two-phase flow), this low-pressure area becomes smaller compared to the case without droplets. As a result, the wing produces less lift, where the lift drops more noticeably. the presence of water droplets reduces the wing's ability to generate lift.

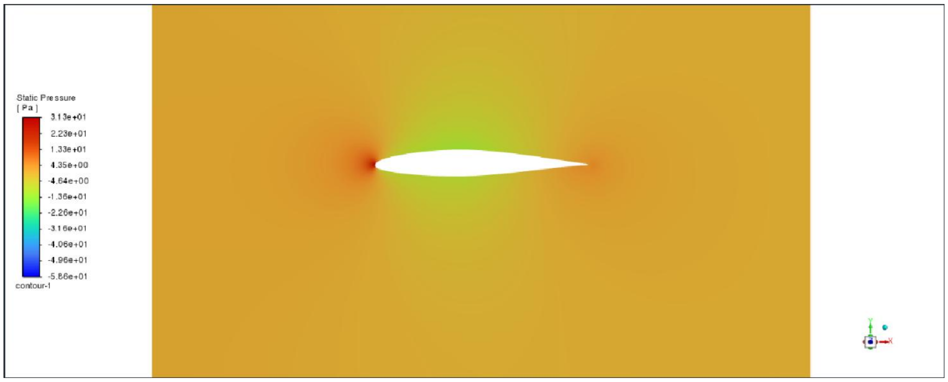

Figure 3. (a) Static Pressure(contour-1)

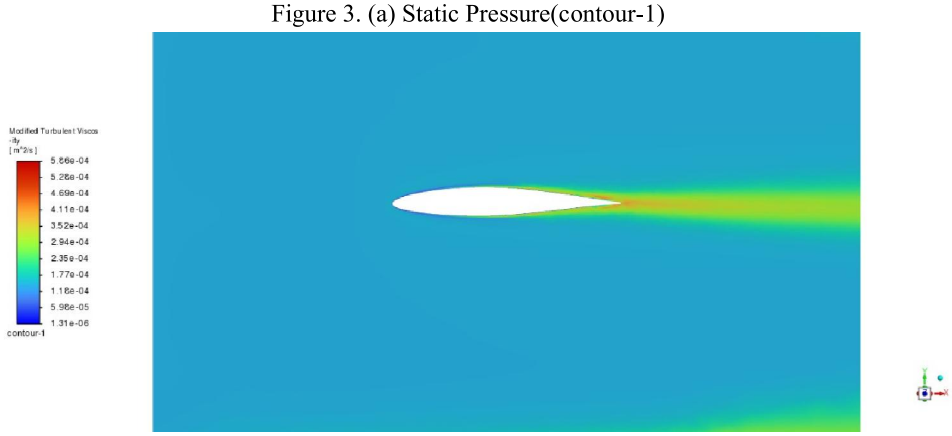

Figure 3. (b) Modified Turbulent Viscosity

The air moving over the top of the wing (where lift is created) reached its fastest speed around 40% of the wing's length from the front. After that, the air began to slow down. The velocity image from the two-phase flow shows that the water droplets caused a thicker area of disturbed air behind the wing—called the wake—compared to when there were no droplets. This thicker wake means more drag and less smooth airflow, showing how the water droplets affect the wing's performance. When two-phase flow (like air mixed with water droplets) moves over the wing, the airflow slows down earlier on both the top (suction side) and bottom (pressure side) surfaces of the wing. Because of this early slowdown, the pressure difference between the top and bottom of the wing becomes smaller than in normal one-phase (just air) flow. As a result, the wing generates less lift, and its efficiency measured by the lift-to-drag ratio also drops when the second phase (like droplets) is present. It was observed that when water droplets were present in the airflow (two-phase flow), the pressure difference between the top and bottom of the wing became slightly smaller compared to when only air was flowing (one-phase flow). Because of this, both the lift and the lift-to-drag ratio were a bit lower with the presence of the water droplets.

DOI: 10.48175/IJARSCT-26350

IJARSCT

International Journal of Advanced Research in Science, Communication and Technology

International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal

ISSN: 2581-9429

Volume 5, Issue 3, May 2025

Impact Factor: 7.67

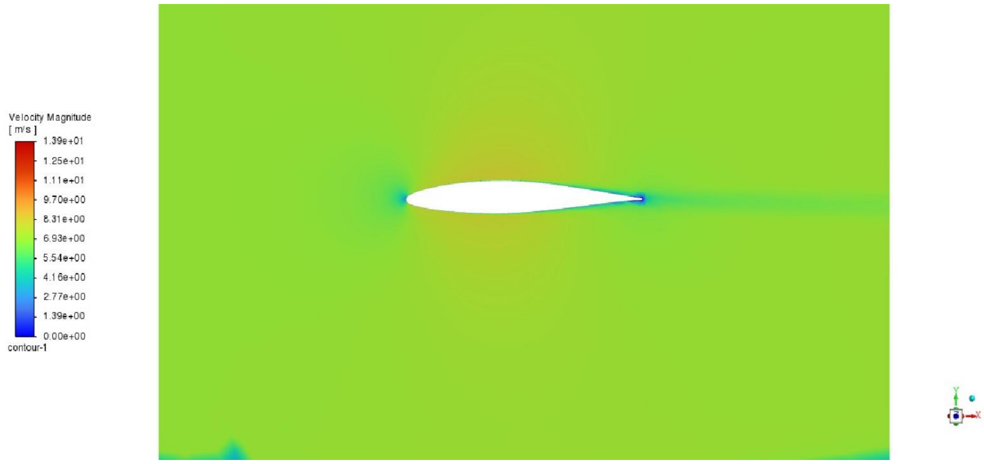

Figure 3.c Velocity Magnitude(contour-1)

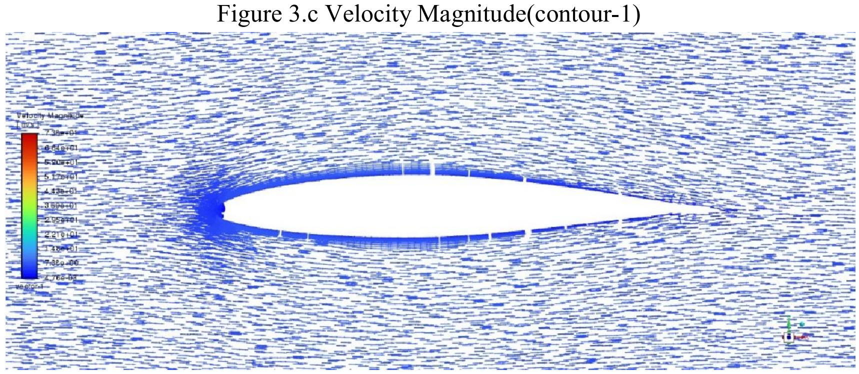

Figure 3.d Velocity Magnitude (vector-1)

The simulation results showed that water droplets tended to break up in areas where the air pressure around the wing was higher.

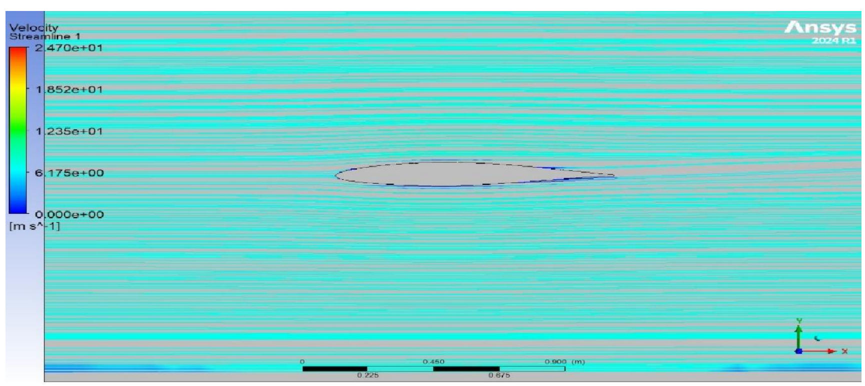

Figure 3.e Velocity streamline

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 5, Issue 3, May 2025
Impact Factor: 7.67

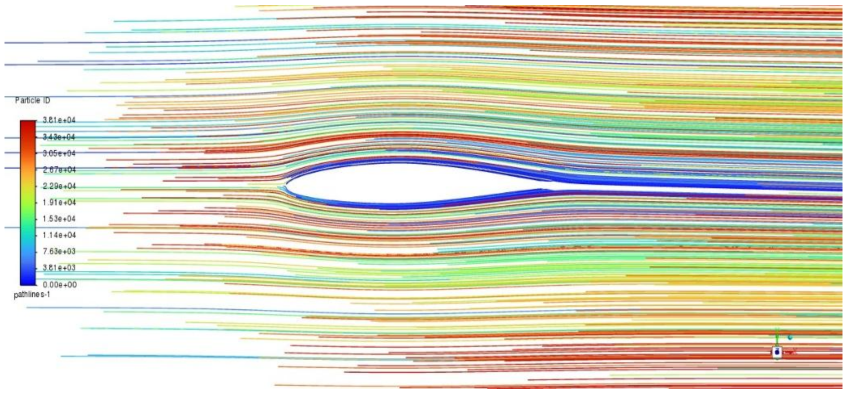

Figure 3.f Pathline s-1

IV. CONCLUSION

This study tested a rectangular wing based on the NACA 641-212 airfoil a full-sized (0.43 m chord). First, they ran simulations in Ansys Fluent at a Reynolds number of 210,000, testing in both dry air (one-phase) and air mixed with water droplets (two-phase).Then they ran real life wind tunnel tests under to check if the simulation results matched reality, the study compared airflow alone (one-phase) with airflow mixed with water droplets (two-phase). It found that the water droplets reduced the wing's aerodynamic performance by about 4%. This drop in performance was seen in simulations, mainly due to changes in pressure around the wing.

REFERENCES

[1]. K. Kamura, K. Toda, and M. Yamamoto, ―Numerical simulation of performance change of airfoil due to sand erosion.‖ J. Nihon Kikai Gakkai Ronbunshu, B Hen/Transactions of the Japan Society of Mechanical Engineers, vol. 67(662), Oct. 2001, pp. 2397-2404. Available: https://doi.org/10.1299/kikaib.67.2397 .

[2]. H.Qu, J.Hu, and X.Gao, The Impact of Reynolds Number on Two-Dimensional Aerodynamic Airfoil Flow,‖ 2009 World Non-Grid-Connected Wind Power and Energy Conf. China, 2009.

[3]. D. C. Douvi, D. P. Margaris, and A. E. Davaris. (Feb. 2017). Aerodynamic performance of a NREL S809 airfoil in an air-sand particle two-phase flow. J. Computation. 5. Available: https://doi.org/10.3390/computation5010013.

[4]. Chen, B.; Wang, L.; Gong, R.; Wang, S. Numerical simulation and experimental validation of aircraft ground deicing model. Adv. Mech. Eng. 2016, 8, 1687814016646976.

[5]. Durst, F.; Milojevic, D.; Schönung, B. Eulerian and Lagrangian predictions of particulate two- phase flows: A numerical study. Appl. Math. Model. 1984, 8, 101–115.

[6]. Endres, M.; Sommerwerk, H.; Mendig, C.; Sinapius, M.; Horst, P. Experimental study of two electro-mechanical de-icing systems applied on a wing section test edinanicing wind tunnel. CEASAeronaut.J. 2017, 8, 429–439.

[7]. Dong, W.; Ding, J. Experimental study on the ice freezing adhesive characteristics of metal surface. In Proceedings of the 51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, Grapevine, TX, USA, 7–10 January 2013.

[8]. Sommerwerk, H.; Horst, P.; Bansmer, S. Studies on electronic impulse de-icing of a leading edge structure in an icing wind tunnel.In Proceedings of the 8th AIAAA tmosphericand Space Environments Conference2016, Washington, DC, USA, 13–17 June 2016.

[9]. Ismail, M.; Yihua, C.; Bakar, A.; Wu, Z. Aerodynamic efficiency study of 2D airfoils and 3D rectangular wing in heavy rain via two-phase flow approach. Proc. Inst. Mech. Eng. Part G J. Aerosp. Eng. 2013, 228, 1141–1155.

[10]. Guo S, Li J Y, Yao W X, Zhan Y L, Li Y F, Shi Y Y. Distribution characteristics on droplet deposition of wind field vortex formed by multi-rotor UAV. PLoS One, 2019; 14(7): e0220024. doi: 10.1371/ journal.pone.0220024.

[11]. M. Drela, XFOIL: An Analysis and Design System for Low Reynolds Number Airfoils. In Low Reynolds Number Aerodynamics. Berlin: Springer-Verlag, 1989, vol 54, pp. 1-12. Available: https://doi.org/10.1007/978-3-642-84010-4_1.

[12]. Dimitra C. Douvi, Eleni C. Douvi, Dionissios P. Margaris Computational Study of NACA 0012 Airfoil in Air-Sand Particle Two-Phase Flow at Reynolds Number of Re=1.76×106, 2019, ISSN: 2454-4116.

[13]. Wan, T.; Pan, S.-P. Aerodynamic Efficient Study under the Influence of Heavy Rain via Two-Phase Flow Approach. In Proceedings of the 27th International Congress of the Aeronautical Sciences, Nice, France, 19–24 September 2010