Modeling and Simulating the Techniques of Computational Fluid Dynamics on an Aircraft Wing Model Employing a Typical Section as the NACA 2412 Airfoil

Ravi M Tilavalli [1], G Veerabhadrappa [2], Mahesh [3]

[1] *Department of Mechanical Engineering, Government polytechnic Harihara, Karnataka, India.*
[2] *Department of Mechanical Engineering, Government polytechnic Kampli, Karnataka, India.*
[3] *Department of Mechanical Engineering, Government polytechnic Aurad, Karnataka, India.*

ABSTRACT

The use of analytics analysis to the resolution of aerodynamic issues in general is often challenging. In order to conduct an analysis of these computational models, either experimental or numerical simulation might be used. The numerical technique, on the other hand, is more desirable since the experimental method requires a significant amount of money to be undertaken. This work illustrates the modeling and simulation methods of a computational fluid dynamics (CFD) issue on a model of an aircraft wing, utilizing a standard section such as the NACA 2412 airfoil. This particular wing model may be selected for use in the experimental design of the future. When doing an analysis of the pressure and velocity distribution on the surface of the wing, ANSYS Fluent is often used. The ANSYS Structural software is also responsible for determining the lift and drag forces. It is also possible to determine the coefficients of lift and drag forces by using the data that is collected when the relative velocity inlet between the airflow and the airfoil varies from 0 to 50 meters per second. The numerical findings that were shown are consistent with those of the theory, which indicates that there is a valid alternative to forecast the aerodynamic properties of the tested wing model in the process of constructing Unmanned Aircraft Vehicles (UAVs).

Keywords: Aircraft Wing, NACA 2412, CFD, Lift and drag forces, Airfoil.

1. INTRODUCTION

The use of airplanes, the quickest form of transportation, has been on the rise recently. Since then, a lot of studies have tried to figure out how to make aircraft that fly faster and safer [1].

A plethora of resources for studying aircraft aerodynamics have been created, many of which rely on computational fluid dynamics (CFD) analysis and numerical techniques. That is crucial while planning the experimental UAVs for use in underdeveloped nations like Vietnam. Urban unmanned aerial vehicles (UAVs) have a wide range of potential civilian and military uses, including but not limited to: weather monitoring, scientific data collection, coastline surveillance, and forest fire prevention.

The study of the forces and moments required for sustained air movement is known as aerodynamics [2], an expanded branch of mechanics. Lift, which acts perpendicular to the direction of flight, and drag, or propellant force, which acts perpendicular to the flight, are the aerodynamic forces that operate on a flying object (see Fig. 1). These pressures are proportional to the relative wind, which is the speed of the flow far ahead of the item.

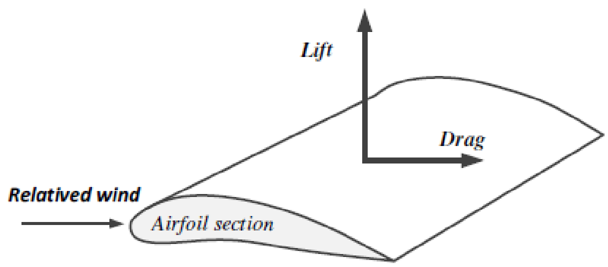


Figure 1. Geometry of an aircraft wing.

Finding out how air interacts with a solid object moving through it, like an airplane wing, is linked to a wide variety of aerodynamic difficulties. on maximize lift and minimize drag on the wing during takeoff and flight, aerodynamic engineers have been studying the best designs for airfoils.

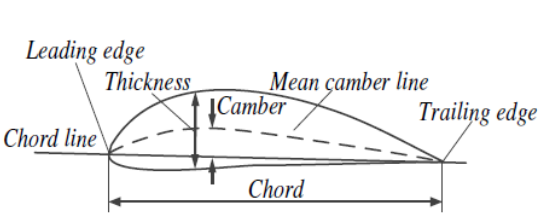
The condition of a wing as seen in cross-area is called an airfoil [3]. Figure 2 depicts the airfoil's fundamental design. National Advisory Committee for Aeronautics (NACA), the precursor to NASA, conducted the first comprehensive analysis of airfoil designs and their operational effectiveness. Airfoil geometry is primarily concerned with the chord, camber, and thickness. Airfoils are normally described by their lift, drag, pressure distribution, and moment about the aerodynamic center, among other performance parameters. Various methods have been used to ascertain the values of these attributes, including wind tunnel experiments, mathematical theory evaluations, and computer models employing CFD simulation tools. The wireless control system was developed to manage and transmit data between the aircraft model and the base station, as mentioned in our group's prior publication [4]. Using the NACA 2412 airfoil, this research conducts computational fluid dynamics (CFD) analyses on a model of an airplane wing using the ANSYS program. The purpose of this research is to evaluate the constructed wing model that might inform the development of future UAV designs.

2. THEORY REVIEW

Lift is the vertical force acting against an aircraft's weight to keep it aloft. While it comes from everywhere in the plane, the wings are the primary source of lift. There is a perpendicular force acting on the item across its center of pressure [5].

See Figure 3 for an illustration of how the flow turning theory is used to show how the airflow around the airfoil produces lift.

A combination of the airfoil's shape, the airflow's viscosity, and the Coanda effect causes the air to pass over the top surface at a vertical velocity, which in turn accelerates the airflow beyond the trailing edge. The airfoil accelerates the air above it as it bends the flow at the top surface, drawing the air down to it. Lift is the result of a low-pressure system that forms above the airfoil as a result of the air's pulling force.



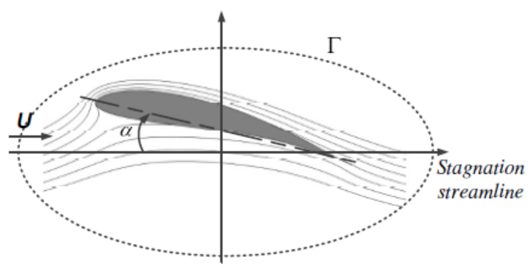


Figure 2. Sketching of airfoil design. **Figure 3. Airflow about the airfoil generating lift.**

The lifting force of an airfoil with a round leading edge and a sharp trailing edge that is submerged in a uniform stream with an effective angle of attack is proportional to the density of the air ρ, the relative velocity of the airflow U, and the circulation G created by the bound vortex, according to the Kutta-Joukowski theorem [7]. This is the definition of the lifting force L that is operating on the airfoil:

$$L = \rho U \Gamma$$

where $\Gamma = \oint_C u.dl$ is proportional to the circulation around the wing.

Similarly, the expression for calculating the drag of airfoil is defined is as follows:

$$D = \frac{1}{2} C_D \rho A U^2$$

where D is the drag force, A is a reference area, and C_D is the drag coefficient.

In a nutshell, the aerodynamic forces exerted by the airfoil are contingent upon the form of the airfoil, the density, viscosity, and compressibility of the air, as well as the surface area of the wing and the angle of attack.

In the realm of aerodynamic forces, the lift coefficient and the drag coefficient are two dimensionless coefficients that are significant. The moving body, the density and velocity of the airstream, and the reference area that corresponds to it are the factors that define these coefficients, which are derived from equations above.

3. MATERIALS AND METHOD

3.1. ANSYS Software

ANSYS provides comprehensive engineering simulation solutions for the design process in various engineering issues. This software is used by companies across several sectors. ANSYS use the Finite Element Method (FEM) together with a range of other programming methods to simulate and optimize diverse design issues. ANSYS consists of many components, among which ANSYS Fluid Flow and Structural are used for conducting simulations. Computational Fluid Dynamics (CFD) is used to analyze issues related to fluid mechanics and dynamics. ANSYS Fluent is a very complete program for CFD modeling that offers advanced physical modeling capabilities and provides rapid and accurate results.

3.2. Description of the geometry model

Figure 4 displays a diagram illustrating the geometric model of the airfoil and aircraft wing. Various numbering methods, such as NACA four digits and five digits, are used to describe the form of an airfoil. The NACA 2412 airfoil [9] is employed in this study for wing design. The first digit indicates the maximum camber as a percentage of the chord length, the second digit represents the position of the maximum camber from the leading edge as a fraction of the chord length, and the last two digits indicate the maximum thickness as a percentage of the chord length [10]. The parameters are selected, namely the airfoil chord $c = 0.3m$ and the airfoil span $l = 1.6m$. The provided dimensions are used in the construction of the experimental wing model. These dimensions are also in agreement with the publicly available data of several test UAV samples in Vietnam.

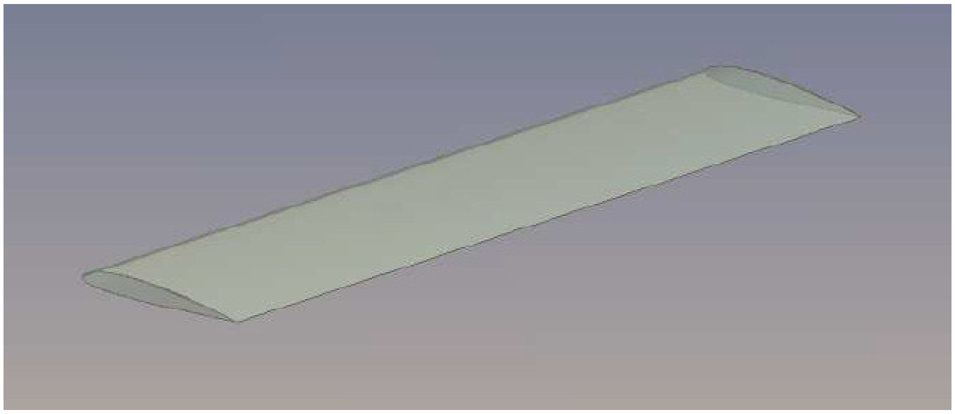


Figure 4. Aircraft wing model created in ANSYS.

3.3. Material selection

The field of metallurgy has been crucial in the advancement of aircraft. In recent times, there has been a use of novel materials in the building of airplanes, including titanium alloy and composites. Nevertheless, these superalloys remain prohibitively costly for anyone constructing airplanes at home. Due to its favorable weight and cost ratio, aluminum alloy continues to be extensively used.

The research incorporates the usage of Aluminum alloy 7075 T6 [11] due to its appealing properties.

The corresponding material characteristics are shown in table 1.

Table 1. Material properties of AA7075-T6

No.	Material Property	Value	Unit
1	Mass density	2810	kg/m^3
2	Young's Modulus	71.7	GPa
3	Shear Modulus	26.9	GPa
4	Poison's ratio	0.33	-

The movement of the fluid is referred to as airflow. The flow characteristics are selected to closely match those employed in the experiment, with a density of $1.225 kg/m^3$ and a kinematic viscosity of $1.7894e^{-5}$. All parameters of the aforementioned materials are used for the purpose of conducting simulations.

3.4. Simulation

Fluid flow analysis involves dividing the flow domain into smaller subdomains, a process known as mesh generation. The mesh is designed to segregate and calculate the characteristics of the fluid flow. Fluent utilizes meshes to represent the fluid domain. Figure 5 displays the mesh that was used. The numerical solution of the Navier-Stokes equations is computed at every node of the mesh. Furthermore, ANSYS Fluent employs an iterative approach to achieve convergence in solving this analysis.

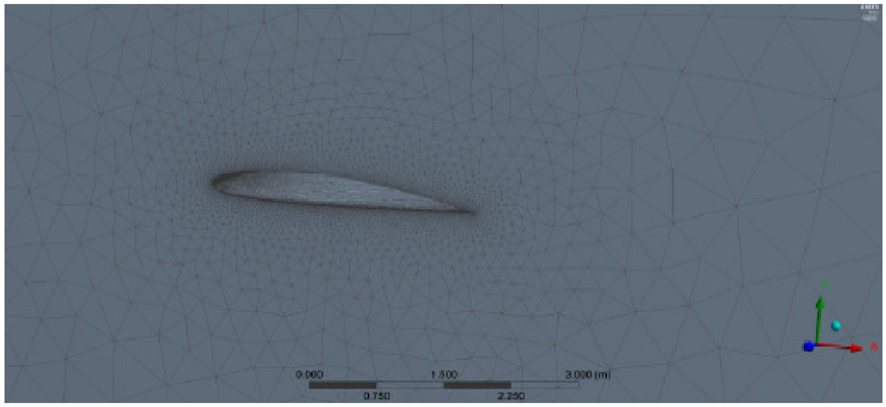


Figure 5. Meshed region.

Prior to initiating the simulation, it was necessary to establish precise parameters and boundary conditions. Initially, it was necessary to create some overarching settings. For instance, gravity is disregarded, time is considered to be constant, velocity is assumed to be in an absolute reference frame, and the solver used is pressure-based. Then, the laminar model is chosen. Additional precise approaches also have to be given. Pressure, momentum, dissipation, and energy are all represented by second order functions in the model.
Higher order functions often provide more accuracy compared to first order approximations, but they also need more time to compute. The next stages include establishing boundary conditions for the various regions of each mesh. These criteria include setting the wall face to have zero velocity, ensuring symmetry on certain faces, and specifying velocity intake and pressure outlet parameters for the fluid.

The flow velocity inlet is modified in several simulations in this research. The velocity is varied between 0 and 50 m/s, with increments of 5 m/s, to match the testing range suitable for UAVs, which typically operate at low speeds. These simulations are performed with an angle of attack of 0 degrees.

Subsequently, the aerodynamic forces are quantified in each simulation to ascertain the coefficients of lift and drag, which can then be compared to theoretical values.

4. RESULTS AND DISCUSSION

The outcomes of the simulation were analyzed at a number of different phases. ANSYS Fluent is capable of delivering quite a few different sorts of graphics, including pressure and velocity distributions, among others. ANSYS Structural, on the other hand, makes it possible to calculate the forces, displacements, stress, and strain that are associated with the wing.

Following the application of a velocity inlet of 25 meters per second, the pressure contours plot in the airflow is shown in Figure 6.

As can be observed, the zones of high pressure present themselves along the leading edge of the airfoil as well as on the bottom surface of the wind turbine. In addition to this, the zone of low pressure is located on the top surface of the airfoil plane. The hypothesis of lift creation is consistent with this analysis, which is correct.

On the other hand, velocity is a feature that is considered to be significant. There is a representation of the velocity magnitude profiles in Figure 7. When it comes to the leading edge and surface of the airfoil, the flow velocity is very close to zero. On the top surface of the airfoil, however, the fact that the fluid accelerates change is readily apparent.

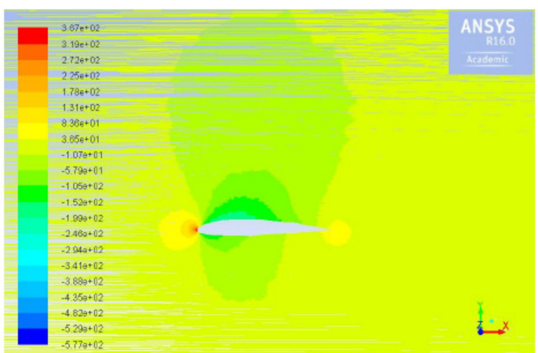


Figure 6. Contours of pressure.

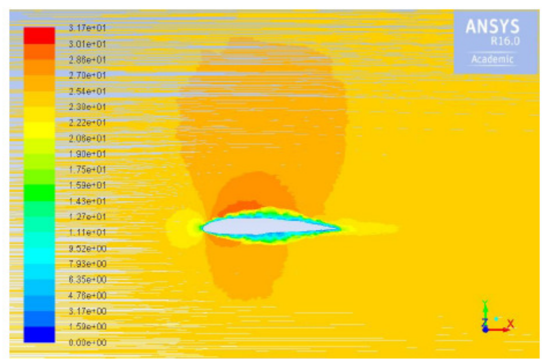


Figure 7. Contours of velocity.

Using ANSYS Structural, the equivalent stress that is placed on the wing is shown in Figure 8. At the point where the section is fastened to the fuselage, the stress is at its highest point. Additionally, lift and drag forces were described as a result of the interaction between the fluid and the structure [12]. It is necessary to gather the force components that match to the velocity inlet. Figure 9 and Figure 10 are two graphs that illustrate the connection between lift and drag in response to relative velocity between the wing and the airflow. These graphs are derived from the data presented in the previous section. A comparison is then made between the outcomes of the simulation and the findings of the theory using equations 2 and 4, with the angle of attack being an infinitesimal 0.025 degrees. The association between these two things is rather strong.

As a consequence, the analytical approach that was suggested has proved that it is a viable option to acquire aerodynamic forces and coefficients by altering the findings that were obtained from the ANSYS simulation. With that being said, it is recommended that more research be conducted in order to lessen the disparities in the outcomes under certain situations and to make it possible to calculate the lift and drag that are associated with friction.

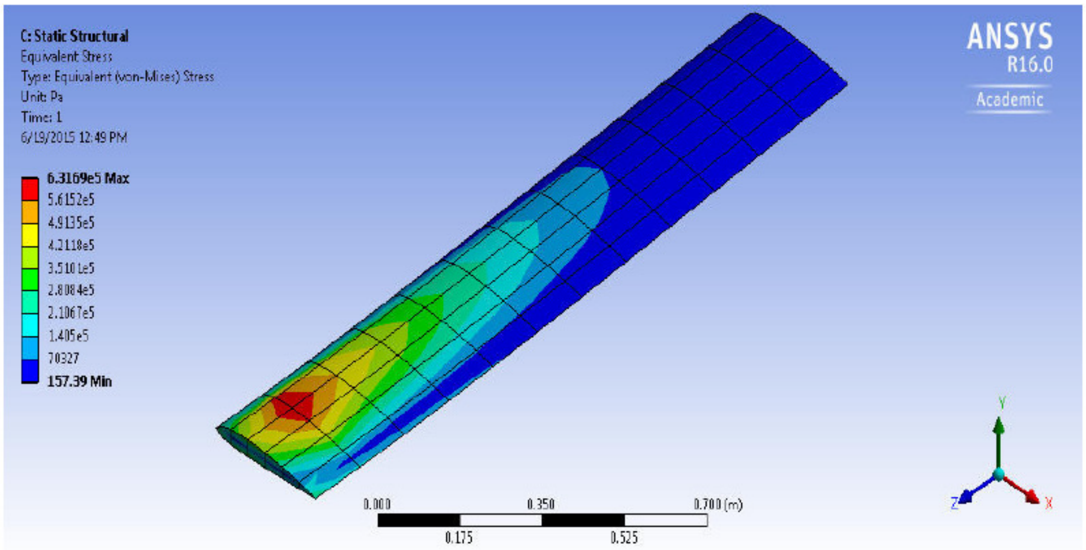


Figure 8. Equivalent Stress in wing.

It is also possible to determine the coefficient of lift and the coefficient of drag for the airfoil model, with the values 0.16 CL = and 0.06 CD =, correspondingly accordingly. There is a correlation between these findings and the hypothesis. It has been shown that the parameters of the setup are appropriate.

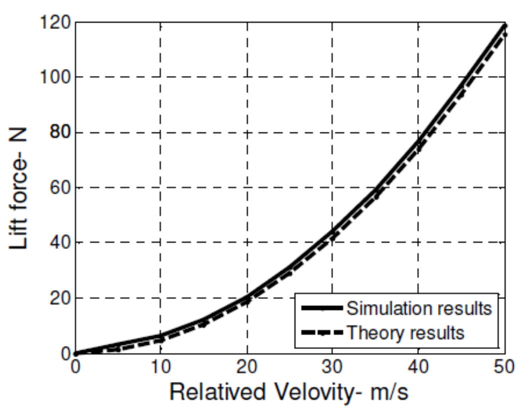


Figure 9. Lift versus relative velocity.

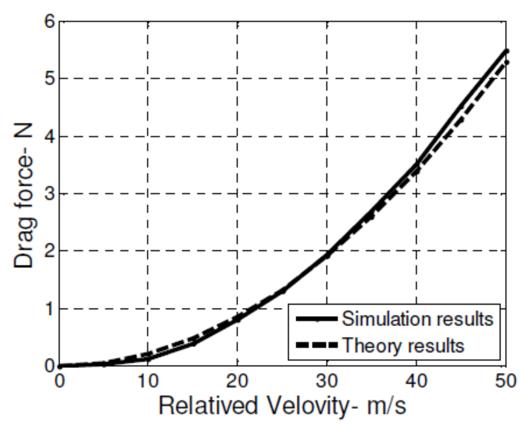


Figure 10. Drag versus relative velocity.

5. CONCLUSIONS

The selected subject of analysis for this study is the aircraft wing model, specifically using the NACA 2412 airfoil. The Fluent and Structural packages of ANSYS Software were used to simulate the model. From an aerodynamic examination of the airflow over an airfoil, the following conclusions may be drawn:

- The pressure on the top surface of the airfoil is lower compared to other areas, and it reaches its highest value at the point of attack.
- Conversely, the velocity of the airflow on the top surface of the airfoil is greater than that on the lower surface. Thus, the simulation approach successfully validated the consistency of lift generation hypothesis. The lift force is about 22.5 times greater than the drag force. It enables the elevation of airborne objects.
- The numerical simulation yielded lift and drag coefficients that closely matched the theoretical values for the NACA 2412 airfoil. This technique is expected to be very relevant for ongoing research and development on the aircraft.
- Ultimately, the consensus is that the airfoil shaped wing would be an excellent choice for producing the next experimental UAVs. The collected data may be used in further research endeavors, such as the selection of sensors and the development of appropriate control systems.

REFERENCES

[1] J.E. Copper, "Towards Faster and Safer Flight Flutter Testing", Proc. Symp. Reduction of Military Vehicle Acquisition Time and Cost through Advanced Modelling and Virtual Simulation, Paris, 2002.

[2] W.Shyy, H. Aono, C. Kang, H. Liu, "An Introduction to Flapping Wing Aerodynamics", Cambridge University Press, pp. 42, 2013.

[3] R.M. James, "The theory and design of two-airfoil lifting systems", Computer Methods in Applied Mechanics and Engineering, vol. 10, pp. 13-43, 1997.

[4] M.T. Nguyen, M.T. Pham, M.C.Vu and D.A. Nguyen, "Design wireless control system for aircraft model", Proceeding of International Conference on Engineering Mechanics and Automation, pp. 283-286, Hanoi, 2014.

[5] Ülgen Gülçat, "Fundamentals of Modern Unsteady Aerodynamics", Springer-Verlag Berlin Heidelberg, pp. 4, 2010.

[6] D. Anderson and S. Eberhardt, "How Airplanes Fly: Physical Description of Lift", The Aviation History Online Museum, 2010.

[7] Wu JC, Lu XY & Zhuang LX, "Integral force acting on a body due to local flow structures", Journal of Fluid Mechanics, vol. 576, pp. 265-286, 2007.

[8] K. Cummings and P. Laws, Understanding Physics, Wiley, 2004.

[9] Prabhakar A. and Ohri A., "CFD Analysis on MAV NACA 2412 Wing in High Lift Take-Off Configuration for Enhanced Lift Generation", J Aeronaut Aerospace Eng., 2: 125. doi:10.4172/2168-9792.1000125, 2013.

[10] Ira H. Abbott and Albert E. Von Doenhoff, "Theory of Wing Sections", Dover Publishing, New York, 1951.

[11] K. Ma, H. Wen, T. Hu, T.D. Topping, D. Isheim, D.N. Seidman, E.J. Laverni, J. M. Schoenung, “Mechanical behavior and strengthening mechanisms in ultrafine grain precipitation-strengthened aluminum alloy”, Acta Materialia, Vol. 62, pp. 141–155, January 2014.

[12] Jin Y., Yuan X., Shin B.R., “Numerical Analysis of the Airfoil’s Fluid-Structure Interaction Problems at Large at Large Mean Incidence Angle”, Proc. ICCFD, Sydney, Australia, 15–19 July 2002.