

Aerodynamic Analysis of Aircraft Wing

Nguyen Minh Triet*, Nguyen Ngoc Viet, Pham Manh Thang

*Faculty of Mechanics and Automation, VNU University of Engineering and Technology,
144 Xuan Thuy, Cau Giay, Hanoi, Vietnam*

Received 09 March 2015

Revised 24 April 2015; Accepted 18 May 2015

Abstract: Aerodynamic problems in general are often difficult to solve by analytics analysis. Experimental or numerical simulation can be used to analyze these computational models. However, due to the large expenses required in the experimental method, the numerical method is more preferred. This paper presents the modeling and simulating processes of computational fluid dynamic (CFD) problem on a aircraft wing model, using typical section as NACA 2412 airfoil. This wing model might be chosen in the future experimental design. ANSYS Fluent is used to analyze the pressure and velocity distribution on the surface of wing. The lift and drag forces are also determined by ANSYS Structural. Additionally, the coefficients of lift and drag forces can be calculated through the data obtained when the relative velocity inlet between the airflow and airfoil changes from 0 to 50 m/s. The numerical results shown are compatible with those of the theory, thus suggesting a reliable alternative to predict the aerodynamic characteristics of the tested wing model in fabricating the Unmanned Aircraft Vehicles (UAVs).

Keywords: Aerodynamic, airfoil, lift and drag, UAVs.

1. Introduction

As the fastest means of transportation available, airplane has been getting more and more popular in recent years. This popularity has led to many research aimed develop faster and safer planes [1]. Many tools based on CFD analysis and numerical methods have been developed and can prove to be extremely useful for the research about the aerodynamic on an aircraft. That is important to design the experimental UAVs in the developing countries, including Vietnam. The UAVs can be used both for military and various civil applications like coastal surveillance, weather observations, forest fire monitoring, scientific data gathering, etc.

Aerodynamic [2] is an extended field of mechanics, which studies the forces and moments necessary to have a sustainable movement in air. Aerodynamic forces acting on flying object are named the lift in the direction normal to the flight and the drag or the propulsive force in the direction

*Corresponding author. Tel.: 84-4-37549667
Email: 4triet@gmail.com

of the flight (see Fig. 1). These forces depend on the flow velocity far ahead of the object, which is called the relative wind.

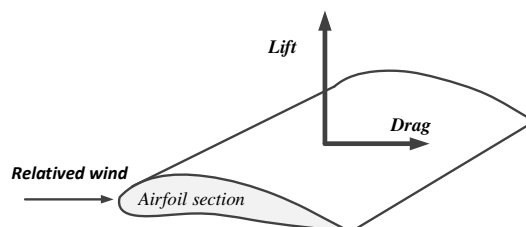


Figure 1. Geometry of an aircraft wing.

A broad range of aerodynamic problems is associated with the determination of the interaction between air and a solid body moving in it, such as an aircraft wing. Aerodynamic researchers have been interested in the optimal shapes of the airfoil, so as to provide the highest lift and the lowest drag to wing during takes off and while in flight.

An airfoil is defined as the state of a wing as seen in cross-area [3]. The basic design of the airfoil is shown in Fig.2. The first systematic study of airfoil shapes and their performances was constructed by the National Advisory Committee for Aeronautics, NACA, (the forerunner to NASA). The chord, camber and thickness are the most important features of airfoil geometry. The performance characteristics of cataloged airfoils typically given include the lift, drag, pressure distribution and moment about the aerodynamic center. Values for these characteristics have been measured by wind tunnel experimentation and are also determined through mathematical theory analyses, or using computational models with CFD simulation tools. In the previous report of our group [4], the wireless control system was designed to control and exchange data between aircraft model and base station. In this paper, CFD analysis on the aircraft wing model using NACA 2412 airfoil are performed by ANSYS software. The goal of this study is testing the fabricated wing model, which might be used for designing the future UAVs.

2. Theory

Lift is the force that directly opposes the weight of an aircraft and holds the aircraft in the air. It is generated by every part of the aircraft, but most of the lift is generated by the wings. This force acts through the center of pressure of the object and is directed perpendicular to the flow direction [5].

The flow turning theory is used to demonstrate airflow about the airfoil generating lift (see Fig. 3). Due to the airfoil geometry, the viscosity of an airflow and the Coanda effect [6], the airflow passes over the upper surface, and creates a vertical velocity of airflow past the trailing edge. As the airfoil bends the airflow near the upper surface, it pulls on the air above it and causes an acceleration of that air down to the airfoil. The pulling of the air causes a low-pressure system to form over the airfoil, creating a force called lift.

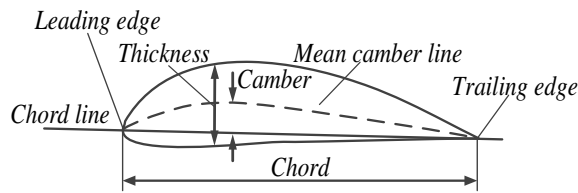


Figure 2. Sketching of airfoil design.

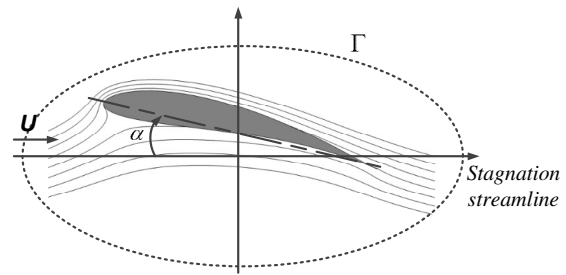


Figure 3. Airflow about the airfoil generating lift.

From Kutta- Joukowski theorem [7], lifting force for an airfoil with round leading and sharp trailing edge immersed in a uniform stream with an effective angle of attack, proportional to the density of air ρ , relative velocity of the airflow U and the circulation Γ generated by the bound vortex. The lifting force L acting on the airfoil is defined as:

$$L = \rho U \Gamma \quad (1)$$

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

If the effective angle of attack is α , the length of the wing l , and the chord length of the airfoil is c , with the Joukowski transformation the magnitude of the circulation is found as $\Gamma = \pi \alpha c U l$. Substituting the value of Γ into Eq. 1 gives the sectional lift force as:

$$L = \pi \alpha \rho c l U^2 = \frac{1}{2} C_L \rho S U^2 \quad (2)$$

where $C_L = 2\pi\alpha$ is called the coefficient of lift, $S = cl$ is the area of the airfoil as viewed from an overhead perspective.

The other aerodynamic force that affects an airfoil and is perpendicular to the lifting force, is called drag. This force opposes the relative motion of the airfoil and has direction parallel to the airflow [8] because skin friction drag appear between the air molecules and the surface of the airfoil.

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

$$D = \frac{1}{2} C_D \rho A U^2 \quad (3)$$

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

In summary, aerodynamic forces of the airfoil depend on the shape of the airfoil, the density, viscosity and compressibility of the air, and the wing surface area and angle of attack.

Lift coefficient and drag coefficient are two dimensionless coefficients that the aerodynamic forces. From Eq. 2 and Eq. 3, these coefficients are determined by the moving body, density and velocity of the airstream, and the corresponding reference area.

$$C_L = \frac{2L}{\rho S U^2} = \frac{L}{qS} \tag{4}$$

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

where $q = \frac{1}{2}\rho U^2$ is the airstream dynamic pressure. Actually, these coefficients can be approximated using the aerodynamic theory as above, numerically calculated or measured in the wind tunnel tests of a complete aircraft wing configuration.

3. Materials and method

3.1. ANSYS Software

ANSYS offers engineering simulation solution sets in engineering problems that a design process requires. Companies in various industries use this software. ANSYS uses FEM and various other programming algorithms for simulating and optimizing various design problems. ANSYS has many sub parts out of which ANSYS Fluid Flow and Structural are chosen to run the simulations. CFD is applied for analysis of fluid mechanics and dynamics problems. The physical modeling capabilities and the fast, accurate CFD results show that ANSYS Fluent is one of the most comprehensive softwares for CFD modeling available in the world today.

3.2. Description of the geometry model

A schematic of the geometry model of the airfoil and aircraft wing is shown in Fig.4. There are several numbering schemes used to characterize the shape of airfoil, as NACA four digits, five digits, etc. In this study, NACA 2412 airfoil [9] is used to design the wing, in which the first digit is the maximum camber in hundredths of the chord, the second digit is the location of the maximum chamber from the leading edge in tenths of the chord, and the last two digits represent the maximum thickness in hundredths of the chord [10]. The parameters are chosen, such as airfoil chord $c = 0.3m$, airfoil span $l = 1.6m$. These dimensions are used to fabricate the experimental wing model, which are also consistent with the open data of a number of test UAVs samples in Vietnam.

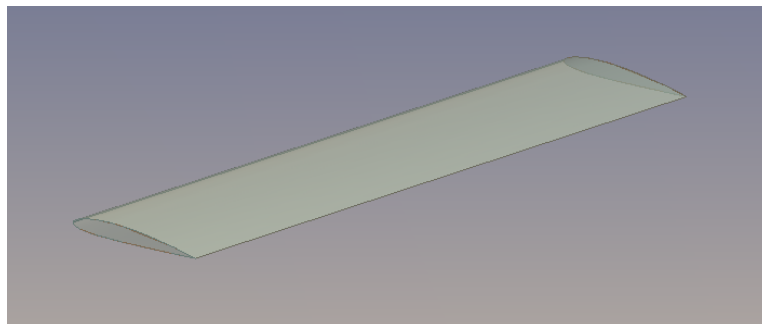


Figure 4. Aircraft wing model created in ANSYS.

3.3. Material selection

Metallurgy has played a key role in the development of aviation. Until recently, some new materials have been applied in aircraft construction, such as titanium alloy, or composite. However, these superalloys are still quite expensive for the aircraft home-builder. With its advantages in weight and cost ratio, aluminum alloy is still used very widely.

Aluminum alloy 7075 T6 [11] is also an attractive material, so it is used in the design of this study. The related material properties are given 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	Poisson's ratio	0.33	-

The fluid flow is considered as airflow. The flow properties are chosen to be similar to that used in the experiment, such as the density is 1.225kg/m³, and the kinematics viscosity is 1.7894e⁻⁵. All parameters of the above materials are applied to set for simulations.

3.4. Simulation

In order to analyze fluid flow, The flow domain is split into smaller sub domains, which is called mesh generation. The intended use of the mesh is to separate and compute the properties of the fluid flow. Fluent uses the meshes to model the fluid space. The mesh used is shown in Fig. 5. It solves the Navier-Stokes equations numerically at each node of the mesh. Moreover, an iterative method is used by ANSYS Fluent to converge on a solution of this analysis.

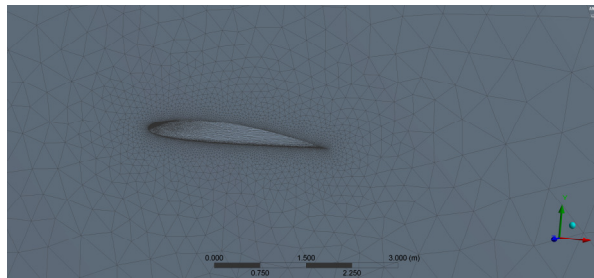


Figure 5. Meshed region.

Before the simulation can be run, specific parameters and boundary conditions had to be set. First some general settings needed to be established. For example, gravity is to be neglected, time to be treated as a steady case, velocity to be taken to be in an absolute reference frame, and the solver used to be pressure based. Next the laminar model is selected. More specific methods also had to be specified. Pressure, momentum, dissipation, and energy are all modeled using second order functions. These higher order functions are generally more accurate than first order approximations, but are also more time consuming. Next steps, boundary conditions are set for the different areas of each mesh, such as wall face with zero velocity, symmetry faces, velocity inlet and pressure outlet for the fluid.

In this project, flow velocity inlet is altered among simulations. Velocity is changed within the range from 0 to 50 m/s with step of 5m/s, in which consistent with the testing range due to UAVs mostly fly under low speed conditions. These simulations are repeated at the angle of attack of 0 degree. After that, aerodynamic forces are measured in each simulation, in order to determine coefficients of lift and drag, and are comparable to theory results.

4. Results and discussion

The simulation results were investigated in various stages. ANSYS Fluent are able to provide several graphic types, such as pressure and velocity distributions. On the other hand, ANSYS Structural allows determining forces, displacements, stress and strain of the wing.

Fig. 6 shows the pressure contours plot in the airflow, when the velocity inlet is applied of 25m/s. As can be seen, the high-pressure regions appear at the leading edge and on the lower surface of airfoil. Besides, the region of low pressure occurs on the upper surface of airfoil. This analysis is accurate with the theory of lift generation.

It is argued that velocity is also an important property. The velocity magnitude profiles are shown in Fig.7. On the leading edge and surface of airfoil, the velocity of the flow is nearly zero. However, the fluid accelerates change clearly on the upper surface of airfoil.

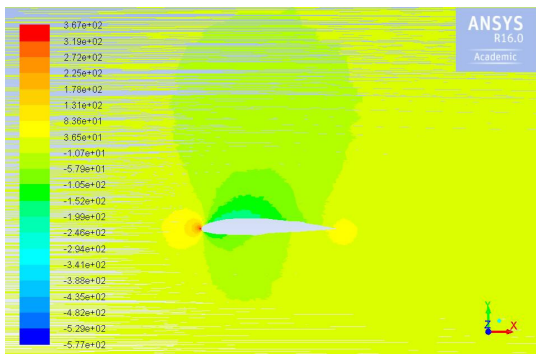


Figure 6. Contours of pressure.

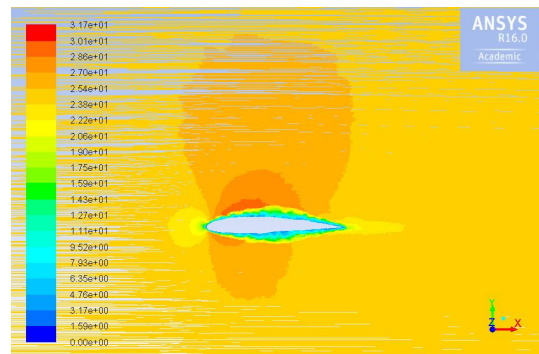


Figure 7. Contours of velocity.

Fig. 8 shows the equivalent stress on the wing using ANSYS Structural. The stress reaches its maximum at the section fixed to the fuselage. In addition, lift and drag forces were defined due to fluid-structure interaction [12]. The force components that which correspond the velocity inlet, are collected. From these data, two graphs of the relationship between lift, drag versus relative velocity between the wing and the airflow are shown in Fig. 9, and Fig. 10, respectively. The simulation results are then compared with theory results by Eq. 2 and Eq. 4, with the infinitesimal angle of attack ($\alpha \sim 0.025$). These comparisons show a good correlation.

Hence the proposed analysis method has demonstrated a workable alternative to obtain aerodynamic forces and coefficients by manipulating the results from ANSYS simulation. However, further investigations are suggested in order to reduce the differences in the results at certain conditions, and to enable calculations of friction related lift and drag.

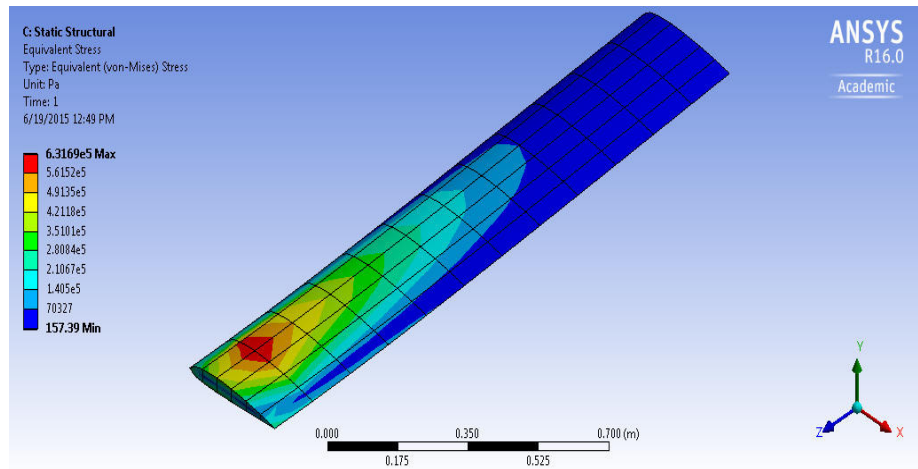


Figure 8. Equivalent Stress in wing.

The coefficient of lift and coefficient of drag for the airfoil model are also defined, where $C_L = 0.16$, $C_D = 0.06$, respectively. These results are comparable to the theory. It is shown that the setting parameters are suitable.

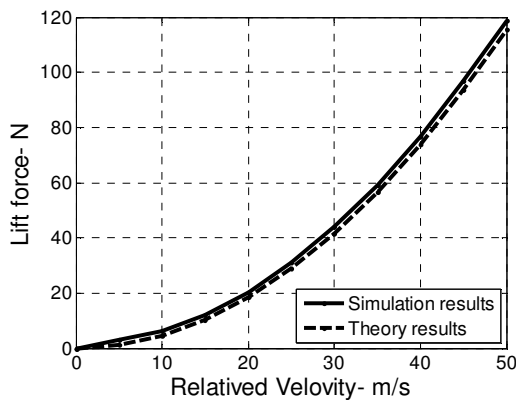


Figure 9. Lift versus relative velocity.

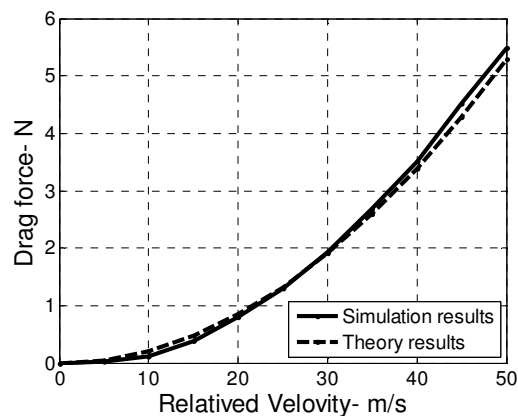


Figure 10. Drag versus relative velocity.

5. Conclusion

In this paper, the aircraft wing model using NACA 2412 airfoil was chosen to be analyzed. Fluent and Structural packages of ANSYS Software were used to simulate the model. Based on aerodynamic analysis of the airflow over airfoil, the following conclusions can be made:

- Pressure is lower on the upper surface of airfoil and reaches its maximum at the point of attack. Meanwhile, the flow velocity on the upper surface is faster than the lower surface of airfoil. Therefore, lift generation theory was demonstrated to be consistent by simulation method.

- Lift force is larger about 22.5 times than drag force. It allows lifting the weight of the flying objects.

- Computed lift and drag coefficients using the numerical simulation were found in good agreement with the theory for NACA 2412 airfoil. This method is likely to be highly applicable to continuing research and development on the aircraft.

- Finally, it is felt that the airfoil shaped wing might be a very good option for manufacturing the experimental future UAVs. The acquired data can assist in future studies such as choosing sensors and designing suitable control system.

6. Acknowledgement

This work is supported by the research group of Mechatronics of Vietnam National University, Hanoi.

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.