Computational Analysis of Fluid Behaviour Around Airfoil With Navier-Stokes Equation

Tulus¹, C Khairani², T.J. Marpaung¹ and Suriati³

¹Department of Mathematics, Universitas Sumatera Utara, Medan, Indonesia
²Undergraduate of Mathematics, Universitas Sumatera Utara, Medan, Indonesia
³Department of Informatics, Universitas Harapan, Medan, Indonesia

Email: tulus@usu.ac.id

Abstract. A study of the development models and shapes of aircrafts component that undergo refinement process over time, must go through the analysis phase. The analysis can be solved by computation method which is supported by the development of numerical method. One important component of aircraft is airfoil. Airfoil is a part that influences the lifting force on the wing of the aircraft. Therefore, in this study should be conducted a study of three different forms of airfoil, about the behaviour of the fluid flow around it, numerically. The approach to fluid flow behaviour can be done using the Navier-Stokes equation. The airflow velocity around the airfoil is the fluid flow behaviour to be analysed, so the numerical solution can be obtained through Finite Element Method (FEM), then it is simulated with COMSOL Multiphysics 5.2a software. Thus, the simulation results will show the interrelations between the angle of attack (α) which varies with the maximum velocity distribution on each airfoil surface, and can be seen the most stable maximum velocity distribution among the three airfoil forms.

1. Introduction

Computational methods are tools that used to solve some problems in life. The development of numerical methods makes the analysis generated by the computational method give a better result. One of the uses of computational method is to analyze dynamic fluid problems. The Navier-Stokes Equation is a series of equations that describe how a fluid flow. The Navier-Stokes equations are commonly used in describing motion of fluids in models relevant to weather, ocean currents, water flow in pipes, and so forth [10]. In simple case, the settlement of the Navier-Stokes equation can be obtained using the science of calculus. One method that can be used to derive numerical solutions from the Navier-Stoke equation is the Finite Element Method (FEM). Using FEM, a complex problem becomes small sections or elements from which a simpler solution can be easily obtained [1].

Thus, with the Navier-Stokes equation can be constructed a simulation with COMSOL Multiphysics 5.2a software. COMSOL Multiphysics 5.2a is software to simulate a physical problem with the finite element method. With this software, researchers usually get a simulation form and numerical results that approach the real results by making some variations. The advantage of making variations in computer simulations and analysis of the main principles of the problem is that users can try various approaches to the solutions to the same problems needed to get the right or correct solutions. [8][12].

In this study should be conducted a study of the fluid flow around airfoil numerically. The result would be approached using the Navier-Stokes equation to be analyzed and the numerical solution could
be obtained through Finite Element Method (FEM), then simulated with COMSOL Multiphysics 5.2a software.

2. Basic Theory

2.1. Fluid flow
Fluid Flow is a substance that changes form continuously when exposed to shear stress. The shear stress is derived from the shear forces divided by the surface area. A shear force is a force component that offends the surface. Some examples of fluids are lubricants, milk, water and air. These substances are categorized into fluids because of the fluid properties that can flow from one place to another [2][10].

2.2. Laminar and turbulent flow
The characteristics of an airflow can be determined by observation. For example, cigarette smoke that flows up, on the near side of cigarettes [12]. The state of the smoke stream looks calm. This fluid flow is called the laminar flow, because the line of current that is formed does not fluctuate or intersect. Laminar flow is relatively low speed. The relatively high flow rate is called turbulent flow. Turbulent flow has a turbulent current line and intersect. An example of turbulent flow is the flow of rafting that occurs in the river. Therefore, there are two basic types of currents, namely Laminar and Turbulent [5].

The Reynolds number is used to determine the characteristics of a stream. When a stream has a low Reynold number, it is a laminar stream. When the Reynolds number becomes larger and passes through the transition the flow can be determined to have turbulent characteristics [3][6].

The forms of Reynold numbers are as follows:

\[ Re = \frac{\rho UL}{\mu} = \frac{UL}{v} \]  

where \( \rho \) is density, \( U \) is speed, \( L \) is characteristic of long, \( \mu \) is dynamic fluid viscosity and \( v(\mu/\rho) \) is kinematic viscosity of fluid.

2.3. Navier-Stokes Equation
Mathematically, the behaviour of a fluid flow can be explained by the Navier Stokes equation. This equation is able to identify forces that occur in a fluid flow. The forces included in it are the force of objects and the force of the surface. Navier Stokes Equation is a differential form of momentum equation. Since the general equation used for the fluid problem is part of the momentum conservation law that is the second law of the newton, where force is the rate of momentum change [3].

\[ \vec{F} = \frac{d}{dt} m \vec{V} \]  

where \( F \) is the force vector, \( m \) is mass, and \( V \) is the velocity vector. If the mass multiplied by the speed it will produce momentum. Then equation (2) is derived into form with aerodynamic variables i.e pressure and density, resulting in an equation called the Navier Stokes equation.

\[ \rho \left( \frac{\partial \vec{V}}{\partial t} \right) + (\vec{V} \nabla) \vec{V} = -p \vec{V} + \mu \nabla^2 \vec{V} \]  

3. Methods

3.1. Settlement with Finite Element Method
The Finite Element Method is one of the numerical methods that is often used to solve problems related to physical phenomena. Usually the form of physical phenomena can be modelled which results in a system of nonlinear equations that is very difficult to solve with analysis. So a numerical method is needed to estimate the results to be obtained. The problems that can be solved by finite element method are divided into two groups, namely group structure analysis and group of non-structural problems. One
of the non-structural problems is the problem of fluid flow [9]. That can be solved using the finite element method.

The popular formulation of Finite Element Method (FEM) used extensively for scientific computing [11]. For the fluid problems can use the equilibrium equation:

\[
[F] = [K][d]
\]

where \([F]\) is force on the node of a column matrix, \([K]\) is stiffness matrix is square matrix and \([d]\) = nodal value in the form of a column

### 3.2. Airfoil Design

Airfoil is a form of an airplane wing that can produce lift or aerodynamic effects when passing through an air flow. The nonmolecular shape of the airfoil is shown in figure 1,

![Non-molecular Airfoil](image)

**Figure 1.** Non-molecular Airfoil.

#### 3.3. NACA

In the early 1930s, NACA - the pioneer of NASA - began a systematic experiment on a series of airfoils that later became renowned as NACA airfoils and are widely used today. NACA airfoil is divided into 'four digits', 'five digits' and '6-series flow laminar airfoil'. Some examples of each type of NACA airfoil are NACA 1412, NACA 23018 and NACA 65-218. The numbers contained in each type of NACA represent the value and location of the camber and the thickness of an airfoil [2]. In this research, we use NACA 2412, NACA 4412 and NACA 6412 to compare the fluid flow rate behaviour of each type of airfoil with variation of angle of attack (\(\alpha\)).

### 4. Result and Discussion

#### 4.1. Basic Equation

Based on the movement of fluid (airflow) around the wings of the aircraft, the type of fluid to be discussed has the properties, incompressible because the air velocity around the wings is considered to be at low speed \((M < 1)\), as well as turbulent flow.

The Navier-Stokes equation used in this study is,

\[
\rho(u, \nabla)u = \nabla \left[ -p + (\mu + \mu_T)(\nabla u + (\nabla u)^T) + F \right]
\]

\[
\rho \nabla.(u) = 0
\]

from the above equation then the boundary value used is:

\[
u = u_0
\]

\[
[-p + (\mu + \mu_T)(\nabla u + (\nabla u)^T)] = -f_0 n
\]

#### 4.2. Geometry Design

On the wing of the plane, there is one part which is capable of producing lift or aerodynamic effect as it passes through an air stream. This section is called airfoil. Airfoil is a 2-dimensional wing form, which is a cross-sectional wing produced by the slings straight wing of the plane. The airflow passing through the airfoil cross section will have different speeds at the top and bottom of the wing. The speed of air flow around this wing, influenced by several factors, one of which is a model or form of airfoil.
Figure 2. Geometry of (a). NACA 2412, (b). NACA 4412 and (c). NACA 6412

4.3. Simulation with COMSOL Multiphysics 5.2a

4.3.1. Simulation on NACA Airfoil 2412

Figure 3. Airflow around the NACA Airfoil 2412 at the (a). Angle of attack 0°, (b). Angle of attack 2°, (c). Angle of attack 4°, (d). Angle of attack 6° and (e). Angle of attack 8°.

From figure 3, the maximum velocity distribution for NACA 2412 is obtained.

Figure 4. The maximum distribution graph around NACA Airfoil 2412

4.3.2. Simulation on NACA Airfoil 4412

Figure 5. Airflow around the NACA Airfoil 4412 at the (a). Angle of attack 0°, (b). Angle of attack 2°, (c). Angle of attack 4°, (d). Angle of attack 6° and (e). Angle of attack 8°.
From figure 5, the maximum velocity distribution for NACA 2412 is obtained.

![Figure 6](image)

**Figure 6.** The maximum distribution graph around NACA Airfoil 4412

4.3.3. Simulation on NACA Airfoil 6412

![Figure 7](image)

**Figure 7.** Airflow around the NACA Airfoil 6412 at the (a). Angle of attack 0°, (b). Angle of attack 2°, (c). Angle of attack 4°, (d). Angle of attack 6° and (e). Angle of attack 8°.

From figure 7, the maximum velocity distribution for NACA 6412 is obtained.

![Figure 8](image)

**Figure 8.** The maximum distribution graph around NACA Airfoil 6412

4.3.4. Comparison of airflow velocity Distribution in NACA 2412, NACA 4412 and NACA 6412

NACA 2412, NACA 4412 and NACA 6412 are 4-digit airfoils released by NACA. The three forms of this airfoil, have a geometry that is almost similar because the chamber is located in the same position that is 40% of the length of the chord and has a thickness of 12% of the length of the chord. This, shown in the last 3 digits of the airfoils, has the same number, which is 412. The distinction of these three forms of airfoil is the large chamber or curvature. Thus, it will be observed the effect of the large difference of curvature on each airfoil to the airflow velocity distribution through the airfoils.
5. Conclusion
The results of the simulation show the airflow velocity distribution in each airfoil that is affected by the angle of attack. With the variation of the attack angle $0^\circ$, $2^\circ$, $4^\circ$, $6^\circ$ and $8^\circ$, it is concluded that:

1. The larger the angle of attack, the maximum air velocity distribution in each airfoil tends to increase.
2. Comparison of the maximum velocity distribution of the three types of airfoil, giving the results of NACA 6412 has the most stable speed distribution.
3. For further research, it can be developed from existing research by analysing other components related to the fluid flow around the wing of aircraft such as pressure, lift coefficient, lift force and others.

Acknowledgments
The authors gratefully acknowledge that the present research is supported by Ministry of Research and Technology and Higher Education Republic of Indonesia and all staff help in the research. The support is under the research grant TALENTA USU of Year 2019.

References
[1] Chao S, Yebin Z, Yusong J 2014 Multibody Finite Element Method and Application in Hydraulic Structure Analysis Mathematical Problems in Engineering 2015 907620.
[2] Tarafder and Nabila 2015 Analysis of potential flow around two-dimensional body by finite element method Journal of Mechanical Engineering Research 7(2), 9–22
[3] Tulus, Suriati, and T. J. Marpaung 2018 IOP Conference Series: Materials Science and Engineering, 4th International Conference on Operational Research Medan 300 012051 (IOP Publishing Ltd).
[4] C. T. Renald, K. Sathiyalingam, V. S. Kumar, S. Shanavasb, and T. Govindasamy 2012 Procedia Engineering, International Conference on Modelling, Optimisation and Computing 38, edited by R. Rajesh, K. Ganesh, and S. C. L. Koh (Elsevier Ltd), pp. 1758–1763.
[5] Tulus, Sitompul OS, Sawaluddin, and Mardiningisih 2004 Mathematical modeling and simulation of fluid dynamic in the continuous stirred tank Bulletin of Mathematics 7(02), p. 1–12 (2004).
[6] D. Liua, Y. Wanga, D. Chena, X. Penga, and Xing X 2012 Procedia Engineering, International Conference on Advances in Computational Modeling and Simulation 31, edited by Z. Lixiang (Elsevier Ltd) pp. 103–109.
[7] Tulus, A. K. Arifin, A. S, and N. Mohamad 2006 Statistics And Applications University Sains Malaysia, Proceedings of the 2nd IMT-GT Regional Conference Of Mathematics (University Sains Malaysia,).
[8] X. J. Yang, D. Baleanu, and J. A. T. Machado, Mathematical Problems in Engineering 2013 769724 (2013).
[9] T. J. Marpaung, Tulus, and S. Suwilo 2018 Cooling Optimization in Tubular Reactor of Palm Oil Waste Processing Bulletin of Mathematics 10(1) 13–24
[10] Tulus, Suriati, and M. Situmorang 2017 *Journal of Physics: Conference Series, International Conference on Computing and Applied Informatics* **801** 012094 (IOP Publishing Ltd).

[11] Tulus and M. Situmorang 2017 *IOP Conference Series: Materials Science and Engineering, 1st Annual Applied Science and Engineering Conference Bandung* **180** 012021 (IOP Publishing Ltd).

[12] Tulus and T.J. Marpaung 2019 *Journal of Physics: Conference Series, The 3rd International Conference on Computing and Applied Informatics 2018* **1235** 012123 (IOP Publishing Ltd).