

# Aerodynamic Analysis Airfoil on an Airplane Wing Using Computational Fluid Dynamic

<sup>1</sup>Yusuf Umardani, <sup>2\*</sup>Ojo Kurdi, <sup>3</sup>Djoeli Satrijo, <sup>4</sup>Aditya Daffa Pambudi, <sup>5</sup>Ian Yulianti

<sup>1,2,3,4</sup>Faculty of Engineering, Department of Mechanical Engineering, Diponegoro University, Indonesia

<sup>5</sup>Physics Study Program, Faculty of Mathematics and Natural Sciences, Universitas Negeri Semarang, Indonesia

\*Corresponding Author's E-mail: [ojokurdi@ft.undip.ac.id](mailto:ojokurdi@ft.undip.ac.id)

**Abstract:** Analysis of the aerodynamic performance of an airfoil cross-section is very necessary to determine the maximum lift force that occurs and the forces acting on the airfoil cross-section such as Drag Force and Lift Force. In this study, a series airfoil cross-section design, a symmetrical airfoil, was tested using Fe software with input of fluid velocity, viscosity and density so that the distribution of velocity and pressure distribution along the airfoil can be known. To obtain maximum performance from this airfoil, different angles of attack are given so that later the maximum angle of attack is obtained to produce maximum lift as well. From the velocity contour and pressure contour read in Fe software along the upper and lower surfaces of the airfoil, the average price is taken and then plotted in a graph to show the magnitude of the lift force and drag force that occurs and from the velocity distribution and temperature distribution, the drag coefficient and lift coefficient prices are obtained. This research is expected to be useful in the world of aerodynamics, especially those related to aircraft wingsso that this modeling can maximize the flight performance of an aircraft and enable the development of aircraft wing designs that are in accordance with the selection of total design with NACA standards.

**Keywords:** Aerodynamics, Airfoil, Lift.

## I. INTRODUCTION

In the aviation world, aerodynamic efficiency is a crucial factor that affects aircraft performance, stability, and fuel consumption. One of the important elements in aerodynamic design is the wing profile or airfoil, which determines airflow characteristics around the wing and generates lift and drag forces. Therefore, the selection and evaluation of the airfoil is an essential stage in aircraft design development.

With the advancement of Computational Fluid Dynamics (CFD) technology, aerodynamic analysis can now be performed numerically using software such as Fe software. This method offers high efficiency in evaluating fluid flow characteristics and pressure distribution around the airfoil, enabling data that closely resembles real conditions without needing physical wind tunnel testing. Through CFD simulation, important parameters such as lift coefficient, drag coefficient, and pressure distribution can be analyzed in detail under various angles of attack.

### 1.1 Background

An aircraft is a combination of various components that work together in an integrated manner to function properly.

Many aspects must be considered by engineers when designing aircraft components, including functionality, safety, cost-effectiveness, and optimal dimensions. Aircraft components are critical parts that require high durability to ensure aircraft safety. One of the aircraft components analyzed in this study is the wing.

Wings, commonly known as airplane wings, are the main components responsible for generating lift to make the aircraft fly. The airplane wing is a vital component that plays a key role in lifting the aircraft body and flying it toward its destination. The fundamental function of an airplane wing is to produce lift through its aerodynamic shape.

Airplane wings have a structure and design that includes an airfoil. An airfoil is the cross-sectional shape of a wing or other surface designed to generate lift when moving through the air. The airfoil shape is very important in determining the aerodynamic performance of the wing or other surfaces.

### 1.2 Research Objective

The topic studied in this research is crossflow. The choice of design has been standardized in many ways, one of which is

the airfoil series. Specifically the airfoil, which is used on the aircraft wing, was selected. The objective of this study is to determine the effect of changes in the angle of attack ( $\alpha$ ) of the airfoil on velocity distribution and pressure distribution, as well as its impact on the lift coefficient (CL) and drag coefficient (CD). The variations of angle of attack used are  $0^\circ$ ,  $3^\circ$ ,  $6^\circ$ ,  $9^\circ$ ,  $12^\circ$ , and  $15^\circ$ , with the maximum lift occurring at a specific angle of attack.

## II. RESEARCH METHOD

Research on airfoil development as a key element in aerodynamics has been conducted extensively in recent years. Various airfoil configurations have been designed based on experimental results to meet specific application needs. Each airfoil has unique characteristics influenced by many factors, making their use specific to certain cases. However, this aspect is often overlooked in model aircraft design due to limited technical knowledge. Therefore, this study focuses on analyzing the aerodynamic characteristics of the airfoil using CFD-based software to understand its performance in an aircraft wing configuration.

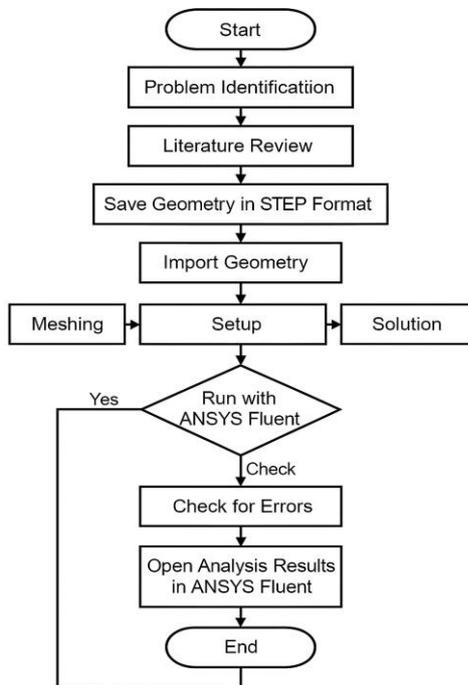


Figure 1: Diagram Alir

### 2.1 Data Analysis

The airfoil profile has a chord length of 180 mm. This airfoil has a maximum thickness of 18% and a maximum camber

located at 15% of the chord.

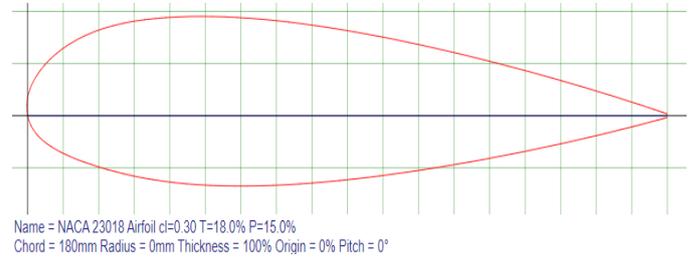


Figure 2: Airfoil

### 2.2 3D Modeling

The 3D model was created using CAD software based on geometry obtained from the manual book and direct measurements during the internship.

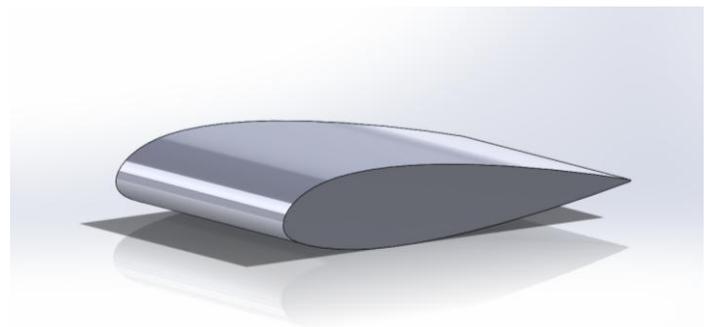


Figure 3: 3D Model in CAD

### 2.3 FE Software Workbench Airfoil

The research method involves performing simulations using Fe software on the airfoil. The steps in this method include:

- a) Importing the airfoil design and generating the 3D model precisely at the origin.

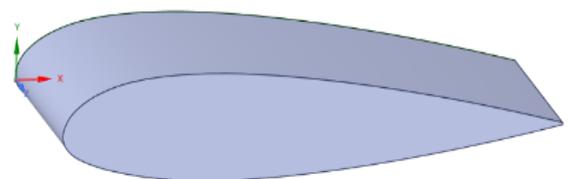


Figure 4: Model 3D Airfoil inFE Software

b) Creating a rectangular enclosure to serve as the airfoil wall.

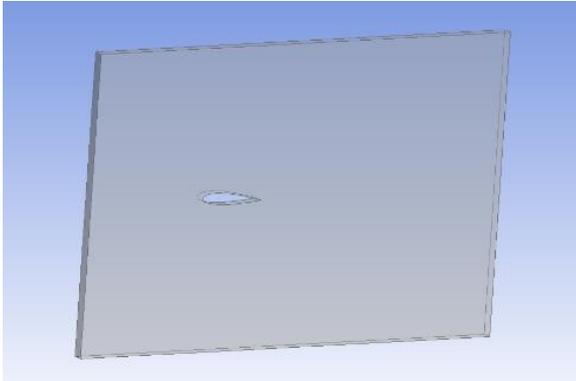


Figure 5: Enclosure for Airfoil

c) Creating a rectangular enclosure at the rear of the airfoil to observe the downstream flow in the tail area.

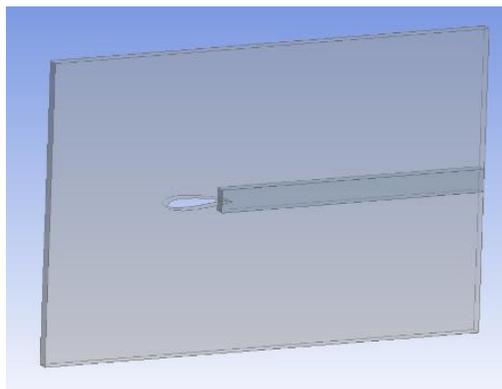


Figure 6: Enclosure Downstream Region

### 2.4 Meshing the Airfoil for FE Software

Meshing is performed to divide the flow domain into small elements used in the numerical computation process.

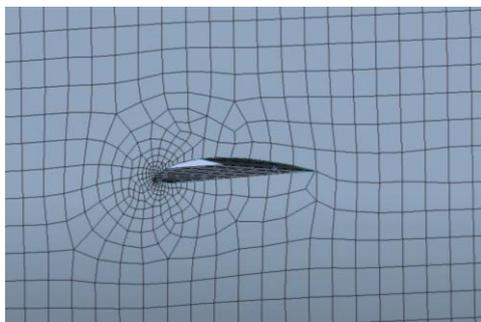


Figure 7: Meshing Downstream Region and Wall

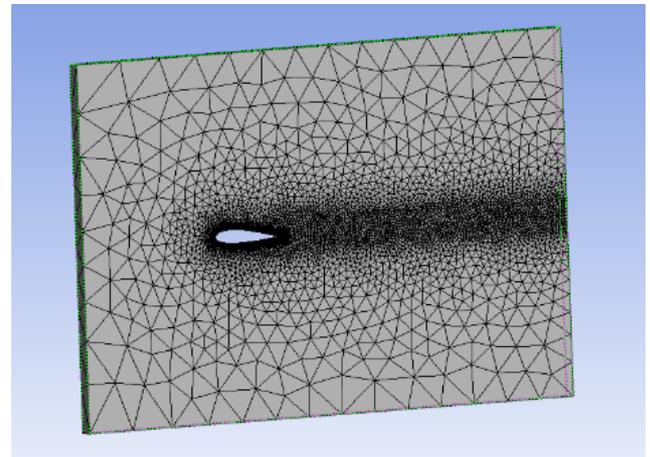


Figure 8: Sizing Mesh Airfoil

### 2.5 Airfoil Setup in FE Software

After the meshing process, the simulation continues by setting up the airfoil in Fe software.

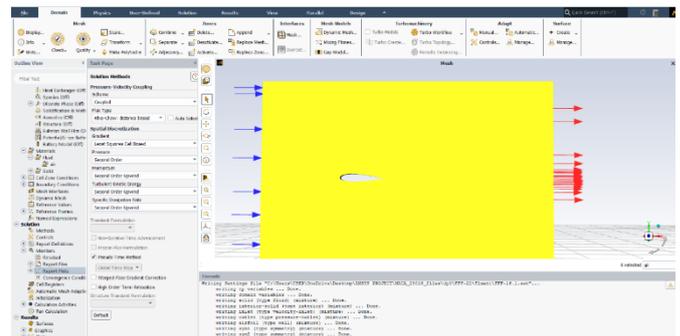


Figure 9: Setup in Fe software Workbench

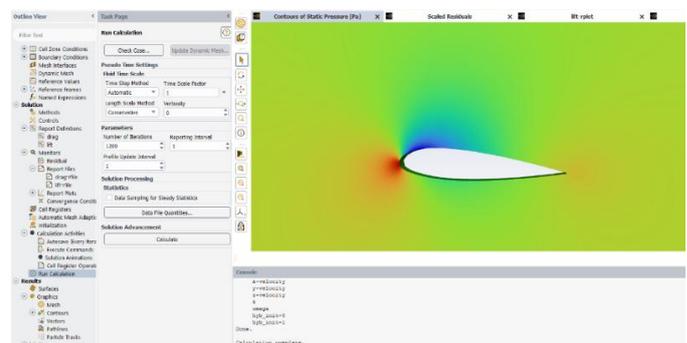


Figure 10: Results

Simulations were carried out for several variations of angle of attack, namely  $0^\circ$ ,  $4^\circ$ ,  $8^\circ$ ,  $12^\circ$ , and  $16^\circ$ .

### III. ANALYSIS AND DISCUSSION

The results of the simulation conducted in this study yielded various visual and numerical data, including pressure contour distributions around the airfoil surface—both on the upper and lower sides. In addition, velocity contour distributions of the fluid flow around the airfoil were also generated, illustrating how the velocity distribution changes due to the geometry of the airfoil and the variation in angles of attack applied. These two types of contours are crucial for understanding fluid flow characteristics, the formation of lift and drag forces acting on the airfoil, and overall aerodynamic behavior.

Table 1: Average Speed and Pressure Values

Angle of Attack	Average Speed		Average Pressure	
	Up (m/s)	Down (m/s)	Up (Pa)	Down (Pa)
0	10.474	13.966	101325.7	101325.7
3	11.863	13.125	101337.6	101403
6	12.867	10.075	101262.4	101409
9	12.572	12.412	101242	101380.4
12	12.110	12.075	101254	101395
15	11.466	12.383	101233.7	101386.7

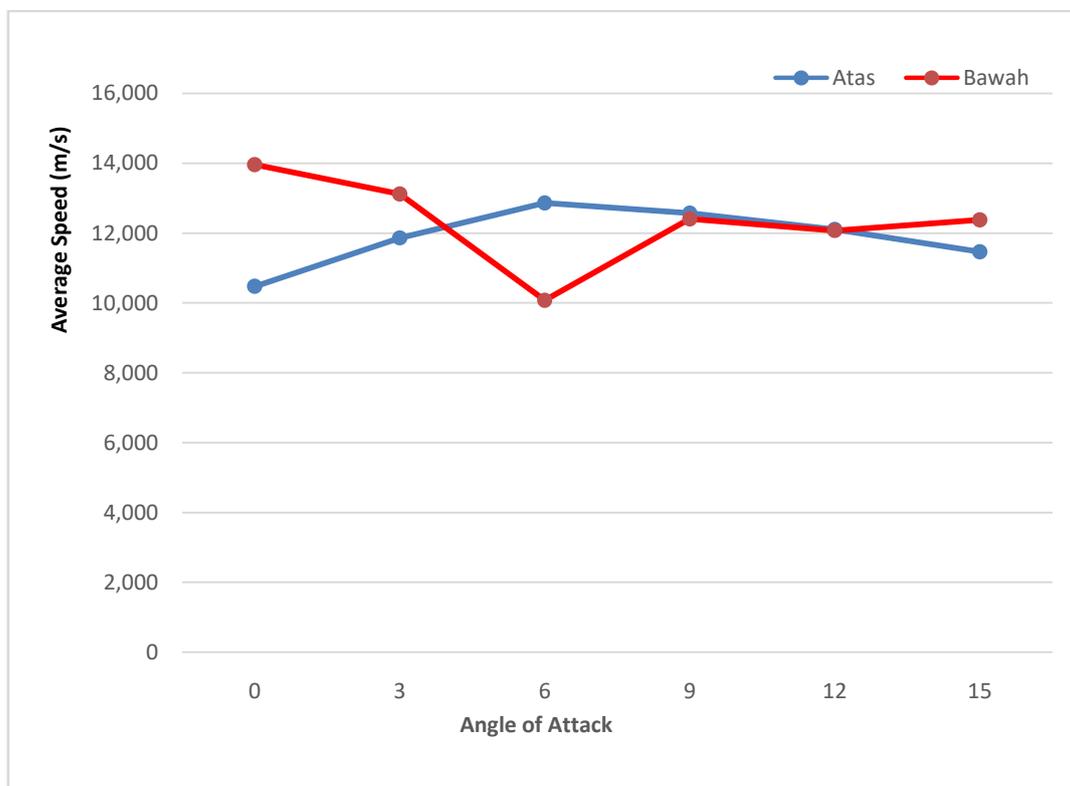


Figure 11: Speed against Angle of Attack

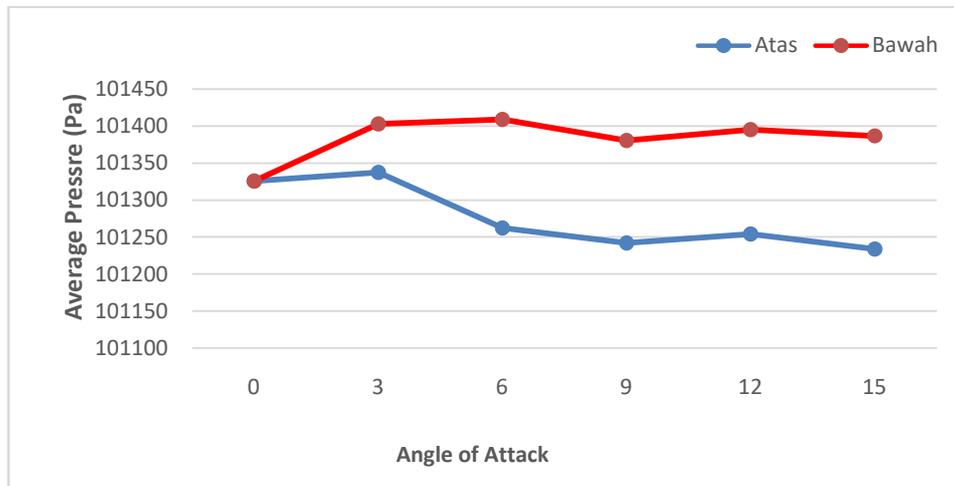


Figure 12: Average Pressure against Angle of Attack

Fe software,  $C_L$  (lift coefficient) and  $C_D$  (drag coefficient) values are two main parameters generated to assess airfoil performance. These values depend on the angle of attack, the geometry of the airfoil, and the flow conditions.

Table 2:  $C_L$  and  $C_D$  Values

Angle of Attack	$C_L$	$C_D$
0	0,097	0.014
3	0,258	0,017
6	0,407	0,028
9	0,465	0,08
12	0,514	0,071
15	0,478	0,059

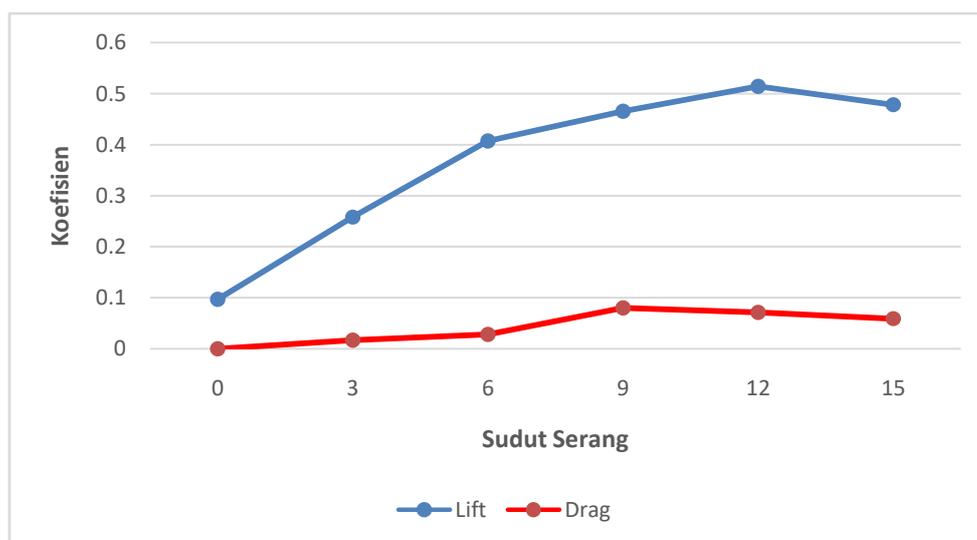


Figure 13: Coefficient Lift dan Drag

#### IV. CONCLUSION

Based on the simulation results of the airfoil with varying angles of attack ( $0^\circ$ ,  $3^\circ$ ,  $6^\circ$ ,  $9^\circ$ ,  $12^\circ$ , and  $15^\circ$ ), it can be concluded that the fluid pressure on the upper surface of the airfoil tends to be lower than on the lower surface, especially from an angle of  $3^\circ$  to  $15^\circ$ . This indicates the formation of a pressure difference that generates lift, while at an angle of  $0^\circ$ , the pressure distribution on both sides is relatively balanced.

Additionally, the fluid flow velocity on the upper surface of the airfoil is higher than on the lower surface at angles of  $6^\circ$ ,  $9^\circ$ , and  $12^\circ$ , which reinforces the pressure difference effect. However, at angles of  $0^\circ$ ,  $3^\circ$ , and  $15^\circ$ , the velocity differences are inconsistent and show varying patterns.

The generated lift increases with the rise in the angle of attack, reaching a maximum value at  $12^\circ$ , which is 33.553 N. Beyond this angle, the lift begins to decrease due to the stall phenomenon. Furthermore, the lift coefficient (Cl) at each angle of attack is always higher than the drag coefficient (Cd), indicating that the aerodynamic performance of the airfoil remains fairly efficient up to the critical angle of attack.

#### REFERENCES

- [1] Harrell, Ghosh, Bowden. (2004). Simulation Using ProModel, *Second Edition, McGraw-Hill Companies*.
- [2] Zero. (2024). The Definition Of Flow Rate. Diakses pada 14 April 2024,
- [3] Anderson, Jhon D., Jr., 2001, Fundamentals of aerodynamics, *McGraw-Hill Book Company, Boston*.
- [4] Ryani, Lenny, and Arie S Ramadhan. "Analisis Geometris Sistem Kendali Elevator Pesawat Udara N219." *METAL: Jurnal Sistem Mekanik Dan Termal* 5, no. 2 (2021): 93.
- [5] E. B. Hamida and G. Chelius, "Strategies for data dissemination to mobile sinks in wireless sensor networks," *IEEE Wireless Commun.*, vol. 15, no. 6, pp. 31–37, Dec. 2008.
- [6] Lennon, Andy, 2005, RC Model Aircraft Design, *Air Age Media Inc., United State of America*.
- [7] NUMERICAL INVESTIGATION OF AERODYNAMIC MasoudJahanmorad Nouri1\*, Habibollah Sayehvand1 and Abolghasem Mekanik, CHARACTERISTICS OF AIRFOIL WITH A GURNEY FLAP, USA.
- [8] Abbott Ira H, Von Doenhoff Albert E andStivers Louis S Jr. (1945), "Summary of Airfoil Data", *AIRFOILRep. No. 824*.
- [9] Bieniawski S and Kroo I (January 2003), "Development and Testing of an Experimental Aeroelastic Model with Micro-Trailing Edge Effectors", American Institute of Aeronautics and Astronautics (AIAA) 2003-220.
- [10] Giguère P, Lemay J and Dumas G (1995), "Gurney Flap Effects Scaling for LowSpeed Airfoils", *American Institute of Aeronautics and Astronautics (AIAA) Paper 95-1881*.
- [11] Kuchemann D (1967), "Inviscid Shear Flow Near the Trailing Edge of an Airfoil", *Z. Flugwiss*, Vol. 15, pp. 292-294.

#### Citation of this Article:

Yusuf Umardani, Ojo Kurdi, Djoeli Satrijo, Aditya Daffa Pambudi, & Ian Yulianti. (2025). Aerodynamic Analysis Airfoil on an Airplane Wing Using Computational Fluid Dynamic. *Current Journal of Engineering and Science Research*. 2(5), 13-18. Article DOI: <https://doi.org/10.47001/CJESR/2025.205003>

\*\*\* End of the Article \*\*\*