Jurnal Rekayasa Mesin
p-ISSN: 1411-6863, e-ISSN: 2540-7678
Vol.20, No.3, December **2025**, p. 435-444

# Performance Test of Airbus A380-like using GAMA Water Tunnel and Numerical Approach

**Syafria Wildan Hadi[1*], Fauzun[2], Ata Syifa Nugraha[1], Feri Hidayatulloh[1], Miftah Hijriawan[1]**
[1]Department of Aircraft Maintenance Engineering, Faculty of Vocational, Universitas Sunan Gresik
Komplek PPM Sunanul Muhtadin, Kertosono, Kecamatan Sidayu, Kabupaten Gresik, Jawa Timur 61153
[2] Department of Mechanical and Industrial Engineering, Faculty of Engineering, Universitas Gadjah Mada
Jl. Grafika No. 2 Kampus UGM, Yogyakarta, 55281, Indonesia

*E-mail: *sw.hadi@lecturer.usg.ac.id*

Submitted: 4-12-2025; Accepted: 24-12-2025; Published: 30-12-2025

***Abstract***

*This study investigates the aerodynamic characteristics of the Airbus A380 model using a combination of water tunnel experimentation and Computational Fluid Dynamics (CFD) analysis. The research aims to analyze the relationship between the angle of attack (AOA) and the resulting lift and drag coefficients, providing insights into aerodynamic performance at low-speed flow conditions. Experiments were conducted using the GAMA Water Tunnel with a 1:416 scaled model of the A380. Lift and drag forces were measured using a load-cell sensor connected to an Arduino microcontroller, while CFD simulations were performed using Computational Fluid Dynamics (CFD) with the k–ω SST turbulence model. The simulation and experimental results were compared to validate the numerical approach. The findings show that both lift and drag coefficients increase with AOA up to a critical point, beyond which stall occurs at approximately 14°. The CFD results closely align with the experimental data, demonstrating strong agreement between the two methods. This research validates the reliability of water tunnel testing for aerodynamic studies and contributes to the development of cost-effective experimental methodologies for aircraft performance evaluation.*

***Keywords:*** *A380-like, Drag Coefficient; Lift Coefficient; Pressure Contour; Velocity Contour*

**Abstrak**

Penelitian ini mengkaji karakteristik aerodinamika model pesawat Airbus A380 melalui kombinasi uji eksperimental menggunakan *water tunnel* dan simulasi *Computational Fluid Dynamics* (CFD). Tujuan penelitian ini adalah menganalisis hubungan antara sudut serang (*angle of attack* / AOA) terhadap koefisien gaya angkat (*lift*) dan gaya hambat (*drag*) untuk memahami kinerja aerodinamika pada kondisi aliran berkecepatan rendah. Pengujian dilakukan di fasilitas GAMA Water Tunnel dengan model berskala 1:416, di mana gaya angkat dan hambat diukur menggunakan sensor *load cell* yang terhubung dengan mikrokontroler Arduino. Simulasi numerik dilakukan dengan metode *Computational Fluid Dynamics* (CFD) dan model turbulensi *k–ω SST*, kemudian hasilnya divalidasi dengan data eksperimen. Hasil menunjukkan bahwa nilai koefisien gaya angkat dan hambat meningkat seiring bertambahnya AOA hingga titik kritis, di mana fenomena *stall* terjadi pada sekitar 14°. Kesamaan tren antara hasil simulasi CFD dan uji eksperimen membuktikan bahwa kedua metode memiliki tingkat kesesuaian yang baik. Penelitian ini menegaskan keandalan pengujian *water tunnel* sebagai metode alternatif yang efektif untuk studi aerodinamika dan pengembangan metodologi eksperimental berbiaya efisien dalam evaluasi kinerja pesawat terbang.

**Kata kunci:** *A380-like, Drag Coefficient; Lift Coefficient; Pressure Contour; Velocity Contour*

## 1. Introduction

Aircraft have become a cornerstone of global mobility, enabling the rapid movement of people and goods across continents. To meet the challenges of fuel efficiency, flight stability, and environmental responsibility, engineers and researchers must possess a comprehensive understanding of aerodynamic behavior. Aerodynamics, as a key discipline in aerospace engineering, governs the interaction between airflow and the surfaces of an aircraft, influencing its performance, energy consumption, and flight dynamics. The study of aerodynamic characteristics particularly lift, drag, and stability forms the basis for designing efficient, stable, and energy-conscious aircraft systems.[1].

Fundamentally, aerodynamics involves the study of how gases move around solid bodies and the forces that arise from this interaction[2]. The lift force allows an aircraft to remain airborne, while the drag force resists its forward motion through the air. The interplay between these forces—along with thrust and weight—governs an aircraft's capacity to achieve and maintain efficient flight[2]. In particular, maximizing the lift-to-drag ratio is essential for enhancing aerodynamic performance, fuel economy, and flight range[3]. A deep understanding of how these forces respond under varying flight conditions serves as the scientific basis for aircraft design, testing, and performance assessment.

Aerodynamic research can be performed through numerical simulations or experimental measurements. Experimental methods, conversely, offer direct physical observation of flow behavior and are indispensable for validating computational models. Among the available experimental facilities, water tunnels have become a cost-effective and practical alternative to traditional wind tunnels, especially for low-speed flow investigations involving scaled models. By matching Reynolds numbers, the behaviour of water flow can accurately replicate airflow under comparable conditions. On the other hand, numerical approaches like *Computational Fluid Dynamics* (CFD) allow detailed visualization and quantification of flow phenomena, including turbulence, boundary-layer behaviour, and separation. However, the reliability of CFD results depends on experimental validation[4].

Numerous studies were conducted using water tunnel testing. Anwar [5] assesses the aerodynamic performance and its flow visualization using a water tunnel to seek a rolled-up vortex, vortex core, and vortex breakdown location that occurred due to the pressure difference on a Sukhoi Su-30MKI-like body. Another similar study that uses a water tunnel, conducted by Sutrisno, which analyses flow visualization, especially to investigate vortex breakdown on some different aircraft models, such as Sukhoi Su-30MKI-like [6], T-50 PAK-FA and F-22 RAPTOR-like[7]. Sutrisno also uses CFD to validate and compare the CFD simulation outcomes against the experimental findings[6]. Assessment on the delta-wing configuration by Wibowo [8] to look for the vortex breakdown around the delta-wing was also using water tunnel.

The Airbus A380 is an aerodyne that represents a significant case study of a superjumbo aircraft with complex aerodynamic characteristics. The generation of aircraft movement utilizes its large fuselage and high-aspect-ratio wings to produce unique aerodynamic behaviors across varying angles of attack (AOA). Analyzing the lift and drag performance of this aircraft provides valuable insights into the effects of geometric configuration on aerodynamic efficiency. The A380 Model is often used as a research object in some research, including from Olejniczak. [9] who analyzes the A380 aerodynamic performance under certain flight conditions, Srikavya [10] who evaluate the contribution of flaps and slats in enhancing lift during takeoff and landing phases, and Reckzeh [11] that explains about detailed aerodynamic design for Megaliner aircraft.

The scientific significance of testing an A380-like model in a low-speed water tunnel lies in the ability to observe complex flow phenomena—such as vortex dynamics and flow separation—which are more pronounced and stable in a high-density medium like water compared to air. Given the A380's massive geometry and high-aspect-ratio wings, understanding these behaviours at low speeds is critical for ensuring stability and fuel efficiency during takeoff and landing phases. Furthermore, this experimental approach provides a high-fidelity benchmark for validating numerical models (CFD), especially in assessing Cl and Cd, offering a cost-effective yet scientifically rigorous methodology for evaluating the performance of large-scale passenger aircraft.

Its aerodynamic performance directly affects its fuel efficiency and stability during flight. Investigating the lift and drag behavior of the Airbus A380 can therefore provide valuable insight into how aircraft geometry influences aerodynamic performance. Furthermore, comparing experimental and numerical results for the A380 can help refine testing methodologies and improve the accuracy of aerodynamic modelling for large passenger aircraft.

## 2. Materials and Methods

### 2.1. Experimental Setup

In this research, we use GAMA water tunnel. This water tunnel has been utilized for various aerodynamic analyses of different aircraft models [5], [6], [7], [8], [12], [13], [14], [15]. It is classified as an open-channel type water tunnel. The tunnel features a test section with a cross-sectional area of 300 mm × 400 mm and is equipped with a water tank of 1,200-litre capacity.

Water from the storage tank was directed through a pipeline into the working section of the tunnel. Before entering the test section, the flow passed through two honeycomb structures designed to maintain laminar flow conditions by minimizing turbulence. The flow rate was controlled using both inlet and exhaust valves, ensuring steady operating conditions during testing. The discharged water was subsequently collected and recirculated back into the storage tank via a pumping system. The flow rate for this research was kept constant at 0,18 m/s to ensure the laminar flow regime for measurement.

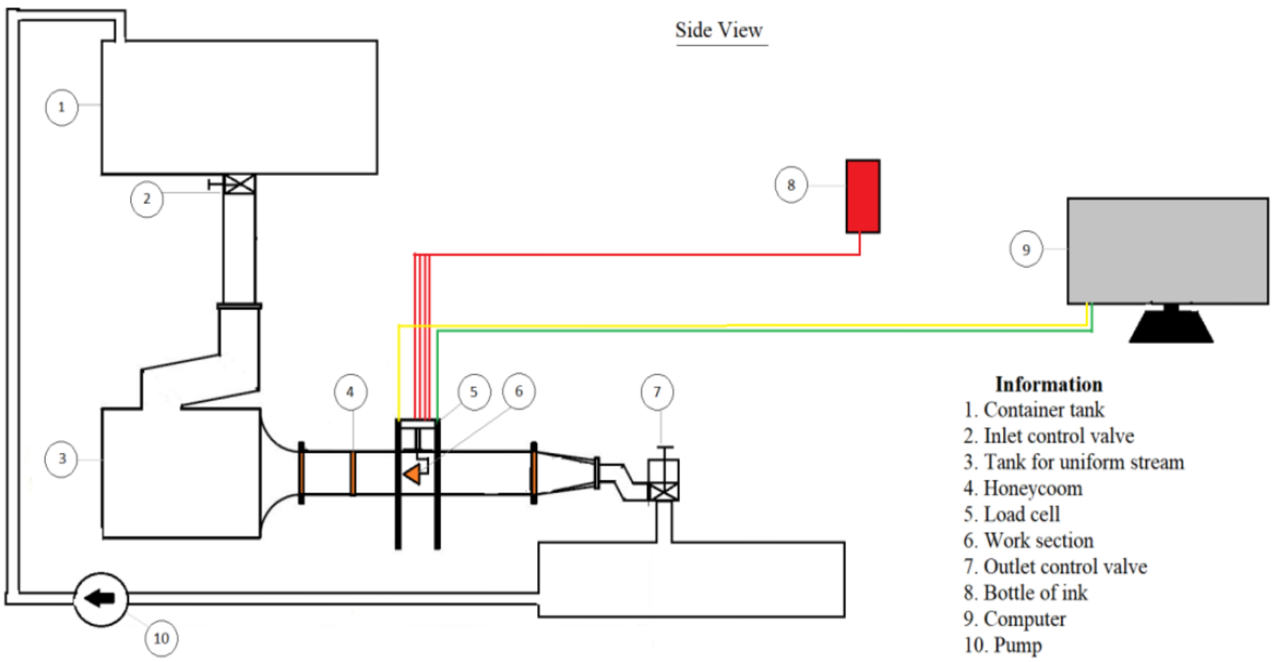



**Figure 1.** GAMA water tunnel test facility [5]

The aircraft model used in this study was based on the Airbus A380, which was redesigned and fabricated using 3D printing technology. The A380-like was then modified using SolidWorks to simplify complex geometrical features. To ensure compatibility with the dimensions of the GAMA Water Tunnel test section, the model was scaled down proportionally to a ratio of 1:416 relative to the actual Airbus A380 aircraft. This scaling produced a wingspan of approximately 18.9 cm, suitable for the tunnel's test cross-section area of 300 mm × 400 mm. The 1:416 scaling ratio was carefully determined to minimize wall interference and ensure the integrity of the measured aerodynamic forces. More importantly, the use of water as the fluid medium is scientifically relevant due to its lower kinematic viscosity compared to air. This characteristic enables the achievement of dynamic similarity (matching Reynolds numbers) at significantly lower flow velocities, ensuring that the results obtained from the scaled model accurately represent the full-scale aircraft's aerodynamic behaviour. Consequently, the interpretation of the lift and drag coefficients in this study is grounded in the principles of dynamic similarity, bridging the gap between laboratory-scale experiments and real-world flight conditions.

A dedicated stainless-steel mounting structure was fabricated to hold the aircraft model securely during experiments. The holder, constructed from a 3.5 mm diameter stainless-steel rod, was designed to provide both mechanical rigidity and minimal interference with the surrounding fluid flow. The holder was attached to the lower fuselage of the model and connected directly to a load-cell system for lift and drag measurement (Figure 2(a)). This configuration ensured stable positioning of the model within the test section while allowing precise data acquisition during test with variations in the angle of attack (AOA). The integrated design between the model, holder, and sensor assembly facilitated accurate force measurements and reliable comparison between experimental and computational results.

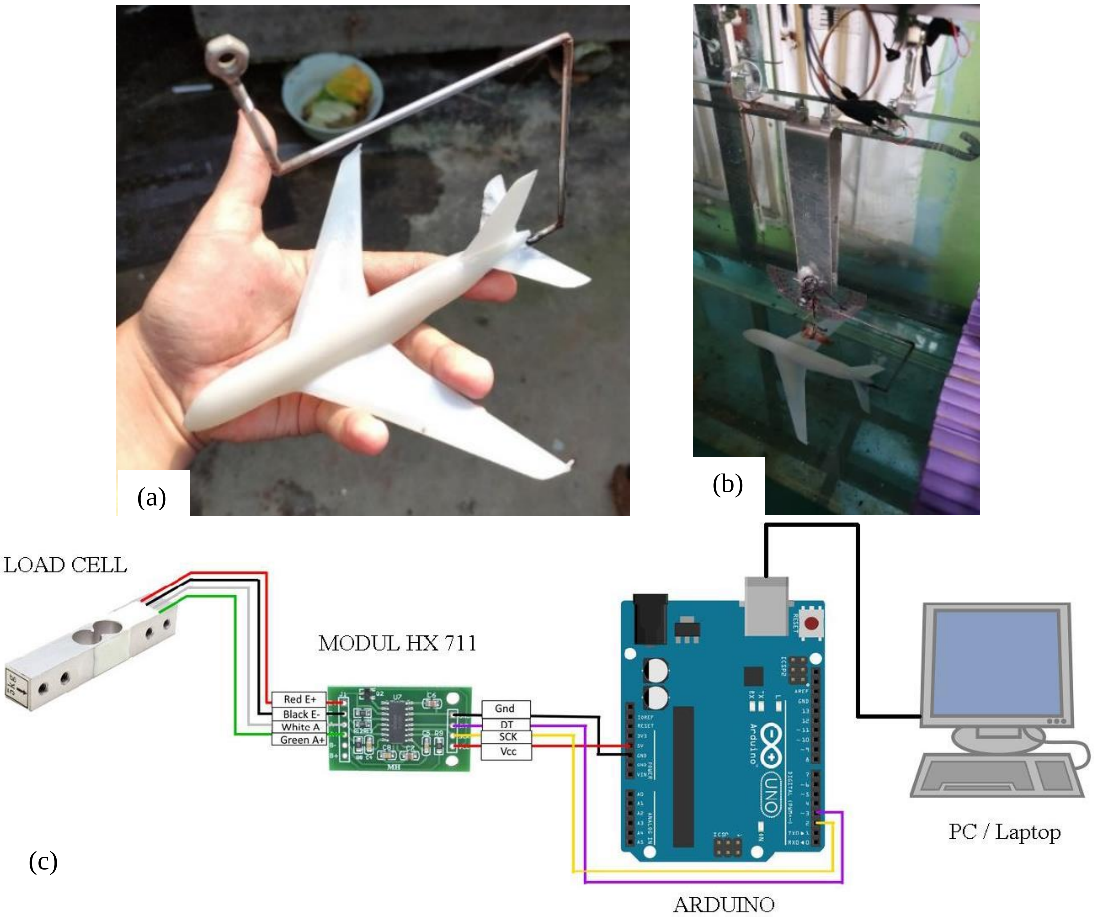



**Figure 2.** Experimental setup: (a) A380-model, (b) A380-model attached to the sensor assembly, and (c) aerodynamic force measurement

Lift and drag measurements were obtained using a load-cell sensor mounted on an aluminium beam that detected deflections caused by aerodynamic forces acting on the model (see Figure 2(b)). The sensor output was amplified via an HX711 module and processed by an Arduino microcontroller for real-time data acquisition through the Arduino IDE (Figure 2(c)). Calibration using known reference weights ensured measurement accuracy, yielding error values of 2.31% for lift and 2.74% for drag, which confirmed the reliability of the sensor system during water tunnel testing. This sensor configuration had been calibrated and had a similar configuration to the sensor from Firmansyah [16].

### 2.2. Numerical Setup

The Airbus A380 model was imported into *ANSYS Fluent* and positioned at the center of the domain to replicate experimental conditions. The computational domain was defined with inlet, outlet, wall, and symmetry boundaries, ensuring accurate flow representation. A structured hexahedral mesh was generated, refined around the aircraft surfaces to capture boundary-layer effects and flow separation accurately. The mesh quality was verified through parameters such as skewness and aspect ratio to ensure numerical stability and solution convergence.

After setting the domain, the computational process was conducted based on Table 1 below. In this research, we employ the k-$\boldsymbol{\omega}$ SST turbulence model. Many CFD-based researchers often use this model because of their ability to handle turbulent flows with high accuracy, especially near the walls.

**Table 1.** CFD setup used in this research

| | | |
|---|---|---|
| ***General*** | *Solver Type* | *Pressure-Based* |
| | *Time* | *Steady* |
| ***Models*** | *Viscous Model* | *SST k-ω* |
| ***Materials*** | *Fluid* | *Water* |
| | *Wall* | *No-slip* |
| ***Boundary Conditions*** | *Velocity_inlet* | *Velocity inlet (Magnitude, Normal to Boundary)*<br>*0,18 m/s magnitude* |
| | *Pressure_Outlet* | *Pressure_Outlet* |
| | *Symmetry* | *Symmetry* |
| ***Solution Initialization*** | *Standard* | *From velocity_inlet* |
| ***Calculation*** | *Number of iterations* | *1000* |
| ***Monitors*** | *Residuals* | *Continuity 0,0005* |

The post-processing phase involved extracting and analyzing simulation results using ANSYS Results and Microsoft Excel. Flow field visualization was performed through contour plots of static pressure and velocity contours across the airfoil sections of the aircraft model. Quantitative results such as the lift coefficient (Cl) and drag coefficient (Cd) were obtained and compared with experimental data from the water tunnel tests. This comparison served to validate the CFD model and assess the consistency between numerical and experimental approaches in predicting the aerodynamic performance of the scaled Airbus A380 model.

## 3. Results and Discussion

### 3.1. Aerodynamic Performance

As you can clearly see in Figures 3(a) and 3(b) show that the data obtained from the CFD method exhibit a trend consistent with that produced by the GAMA Water Tunnel experiments. Both methods reveal an increase in the drag coefficient (Cd) with rising angles of attack (AOA), as well as the presence of a maximum lift coefficient (Cl) within a certain AOA range, followed by a stall phenomenon at higher AOA values. The maximum value of the lift coefficient was obtained in AOA = 14°, and the drag coefficient was increasing as AOA was also going up. Therefore, it can be concluded that the readings captured by the load-cell sensor and processed through the Arduino microcontroller show strong agreement with the results obtained from numerical simulations, as indicated by the similar Cl and Cd curves generated by both the experimental and computational approaches. The observed discrepancies in numerical values between the two methods are likely attributed to variations in mesh density distribution within the CFD model.

p-ISSN: 1411-6863, e-ISSN: 2540-7678
Vol.20|No.3|435-444|December|2025

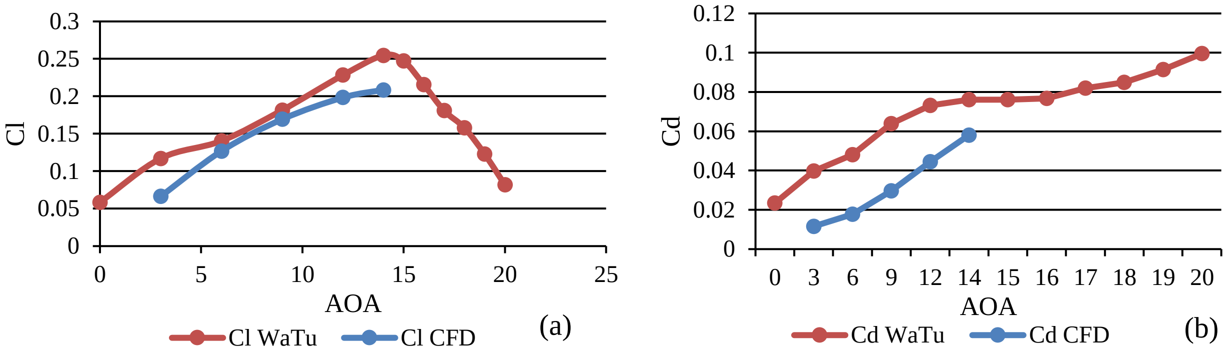



**Figure 3.** Aerodynamic parameters, (a) Lift coefficient and (b) Drag coefficient

3.2. Flow Visualization

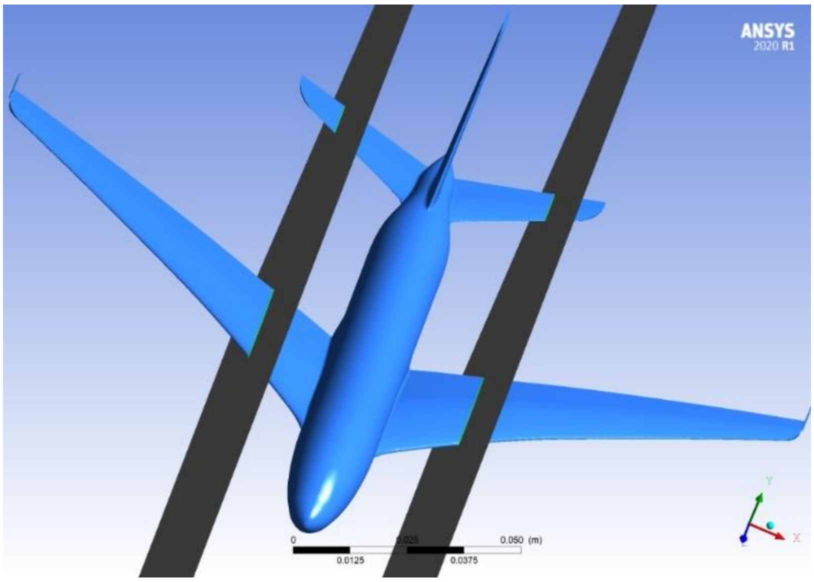
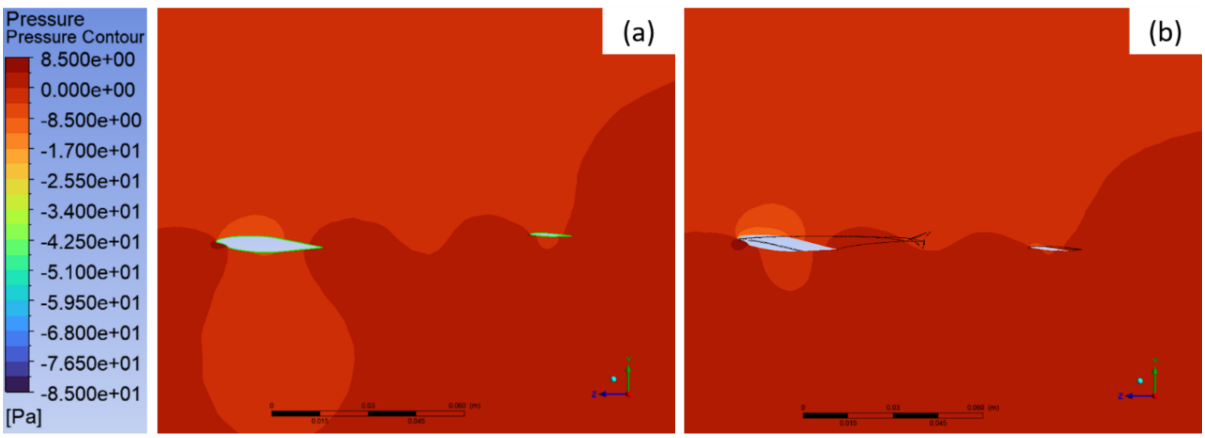



**Figure 4.** Location of sliced cross-section

Flow visualizations were initiated using CFD-post to compare the aerodynamic parameters around the A380-like. The parameters included pressure contours and velocity contours analysis, which located a representative wing cross-section (Figure 4) at varied AOA. The sliced cross-section is 2,5 cm from the symmetry.



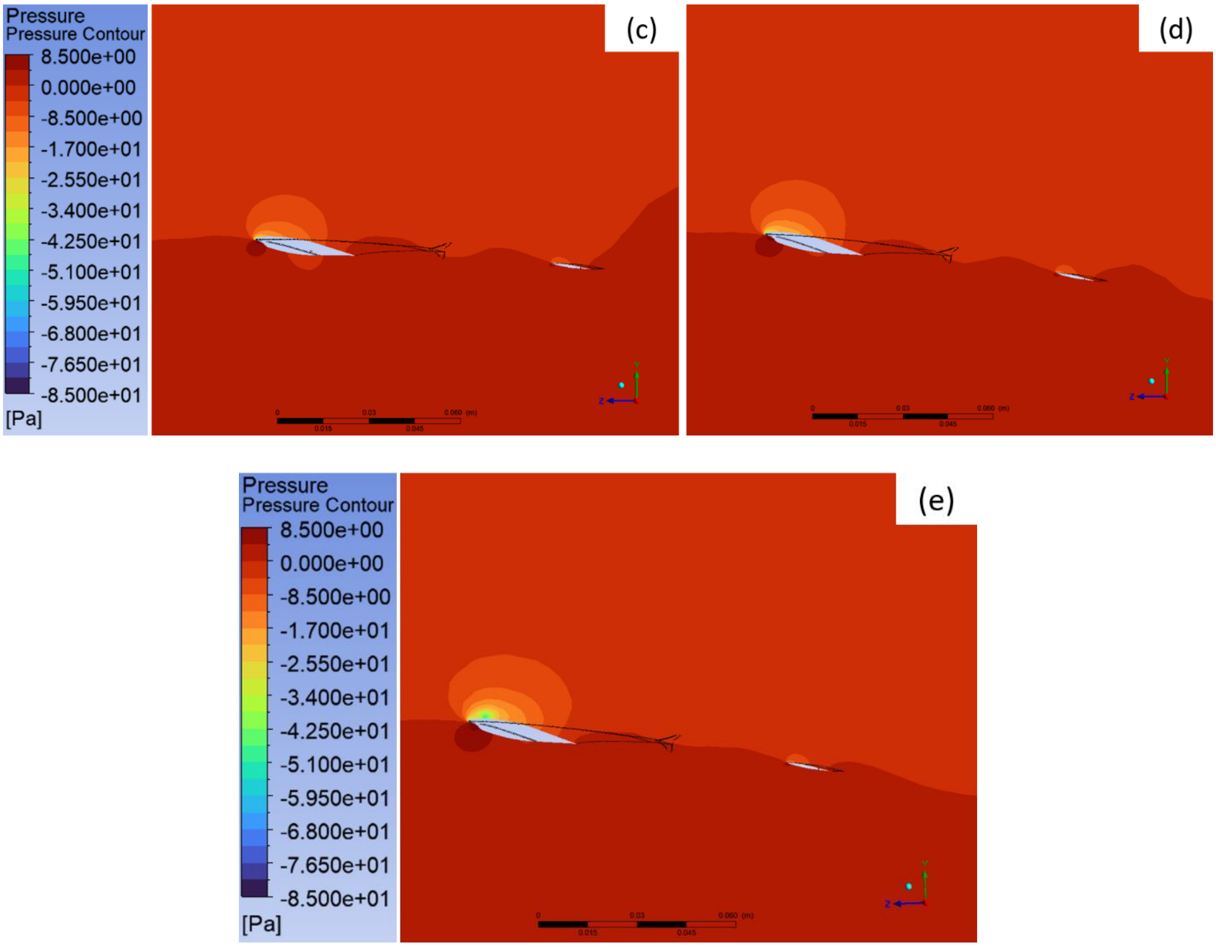



**Figure 5.** Pressure contours at various AOA, (a) 3°, (b) 6°, (c) 9°, (d) 12°, (e) 14°

As shown in all pressure distribution contour plots (Figure 5 (a) – (e)), the pressure on the upper surface of the aircraft wing is generally lower than that on the lower surface, resulting in the generation of lift. At a low AOA of 3° (Figure 5 (a)), the suction peak is concentrated near the leading edge of the upper wing surface. As the AOA increases steadily to 12°, the region of low static pressure expands significantly towards the mid-chord, reflecting the higher lift generation and minimal drag generation as depicted earlier in Figure 3 (b). The negative pressure gradient close to the trailing edge, however, becomes more noticeable at higher AOA, as pictured in Figure 5 (c) to 5 (e). This behaviour is indicated by the increasing pressure difference between the upper and lower surfaces of the wing, both on the front and rear sections of the aircraft.

The velocity contour analysis in Figure 6 (a) – (e) reinforced these findings by illustrating how flow separation developed progressively near the trailing edge as the angle of attack (AOA) increased. At low angles of attack (3° to 9°), which are represented in Figure 6 (a) to 6 (c), the velocity contours exhibit a streamlined flow pattern with a well-defined stagnation point at the leading edge. As the AOA increases gradually, the flow over the upper surface of the A380-like model undergoes significant acceleration, as evidenced by the expanding high-velocity regions in the CFD visualization. This acceleration induces a corresponding drop in static pressure on the upper wing surface, as portrayed in Figure 5 (a) to 5 (c), creating a more pronounced suction peak near the leading edge.

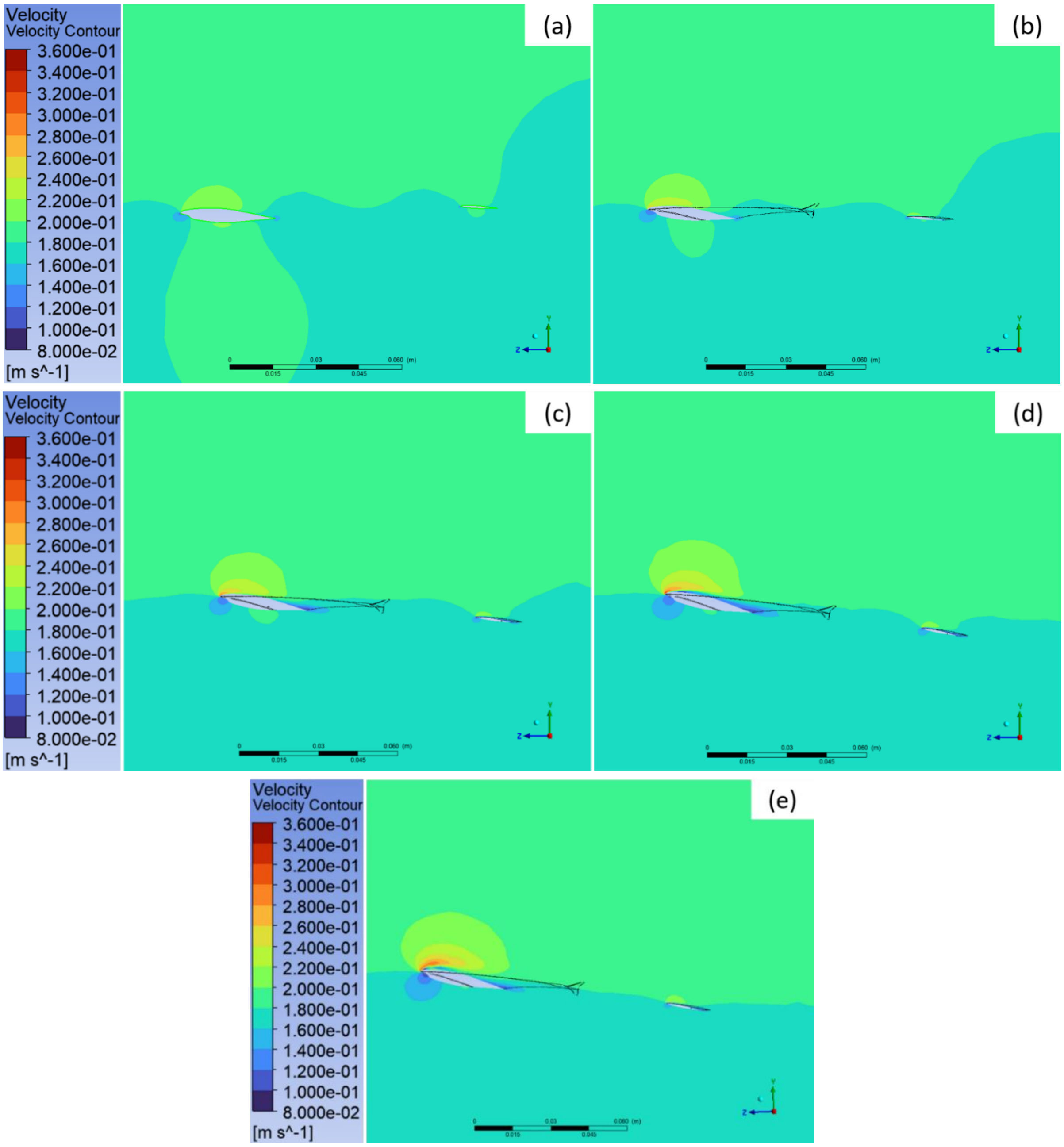



**Figure 6.** Velocity contours at various AOA, (a) 3°, (b) 6°, (c) 9°, (d) 12°, (e) 14°

At moderate AOA values (around 12°–14° in Figure 6 (d) and (e), small separation zones began to appear near the rear portion of the wing. When the AOA exceeded this range, the separated region expanded toward the mid-chord, marking the onset of stall. The velocity contours reveal a noticeable thickening of the boundary layer, followed by the onset of flow separation starting from the trailing edge and migrating forward. As a result, the wake region behind the aircraft became increasingly pronounced, accompanied by stronger turbulence and higher energy losses, which together led to a noticeable increase in drag.

## 4. Conclusion

The experimental and computational analyses conducted in this study successfully demonstrated the aerodynamic features of the Airbus A380 model through both water tunnel testing and CFD simulation. The results showed that the lift coefficient (Cl) increased with the angle of attack (AOA) up to a critical point of 14°, after which stall occurred, while the drag coefficient (Cd) continued to increase across the range. The strong agreement between experimental and numerical results confirmed the reliability of water tunnel testing as a method for assessing aerodynamic performance at a laboratory scale.

**Acknowledgement**

The authors would like to express their gratitude to the Department of Mechanical Engineering, Faculty of Engineering, Gadjah Mada University, for providing access to laboratories and institutional support that facilitated this research.

**References**

[1] A. Magrini, D. Buosi, F. Poltronieri, E. De Leo, and E. Benini, “CFD-Based Analysis of Installed Fuel Consumption and Aerodynamics of Transonic Transport Aircraft during Cruise Flight,” *Energies (Basel)*, vol. 16, no. 8, Apr. 2023, doi: 10.3390/en16083323.

[2] J. D. . Anderson, *Introduction to flight*. McGraw Hill, 2012.

[3] D. H. Neuhart and M. N. Rhode, “Water-Tunnel Investigation of Concepts for Alleviation of Adverse Inlet Spillage Interactions With External Stores NI A,” Washington DC, 1990.

[4] B. B. Gunawan, V. S. Neary, and R. G. Coe, “Model Validation Using Experimental Measurements From The Garfield Thomas Water Tunnel At The Applied Research Laboratory (ARL) At Penn State University,” 2014. [Online]. Available: https://www.researchgate.net/publication/316319841

[5] Z. Anwar, S. Sutrisno, B. W. Setyawan, R. S. Firdaus, and F. N. Wega, “Performance and flow visualization of Sukhoi Su-30MKI like model using GAMA water tunnel,” in *IOP Conference Series: Materials Science and Engineering*, Institute of Physics Publishing, Dec. 2018. doi: 10.1088/1757-899X/434/1/012174.

[6] Sutrisno, T. A. Rohmat, S. B. Wibowo, and S. Iswahyudi, “Vortex dynamics study of the canard deflection angles’ influence on the Sukhoi Su-30-like model to improve stall delays at high AoA,” *Aerospace*, vol. 6, no. 2, Feb. 2019, doi: 10.3390/aerospace6020012.

[7] Sutrisno, S. Bekti Wibowo, S. Iswahyudi, D. Fernando, and Handika, “Comparison of Sukhoi T-50 PAK-FA and F-22 Raptor-Like Fighters Model Flow Visualization Using Gama Water Tunnel,” *International Journal of Mechanical Engineering and Technology (IJMET)*, vol. 10, no. 06, pp. 303–311, 2019, [Online]. Available: http://www.iaeme.com/IJMET/index.asp303http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=10&IType=6http://www.iaeme.com/IJMET/issues.asp?JType=IJMET&VType=10&IType=6

[8] S. B. Wibowo *et al.*, “An investigation into the use of GAMA water tunnel for visualization of vortex breakdown on the delta wing,” in *AIP Conference Proceedings*, American Institute of Physics Inc., Aug. 2018. doi: 10.1063/1.5049998.

[9] D. Olejniczak and M. Nowacki, “Assessment of the selected parameters of aerodynamics for Airbus A380 aircraft on the basis of CFD tests,” in *Transportation Research Procedia*, Elsevier B.V., 2019, pp. 839–846. doi: 10.1016/j.trpro.2019.07.118.

[10] B. P. Srikavya and A. S. Rao, “International Journal Of Research In Aeronautical And Mechanical Engineering Aerodynamic Cfd Analysis On High-Lift Multi-Element Wing Of Airbus A380,” *International Journal Of Research In Aeronautical And Mechanical Engineering*, vol. 2, no. 3, pp. 175–186, 2014.

[11] D. Reckzeh, “Aerodynamic design of the high-lift-wing for a Megaliner aircraft,” *Aerosp Sci Technol*, vol. 7, no. 2, pp. 107–119, Mar. 2003, doi: 10.1016/S1270-9638(02)00002-0.

[12] S. B. Wibowo *et al.*, “Vortex dynamics study and flow visualization on aircraft model with different canard configurations,” *Fluids*, vol. 6, no. 4, Apr. 2021, doi: 10.3390/fluids6040144.

[13] S. B. Wibowo, D. P. Lufstansa, D. Adiyantoro, and R. Safarudin, “Visualisasi Aliran Pada Model Pesawat Tempur Untuk Variasi Sudut Canard Posisi Sejajar Sayap Utama Menggunakan Water Tunnel,” 2019.

[14] S. B. Wibowo, Sutrisno, T. A. Rohmat, S. Siswantoro, D. Fernando, and W. F. Naufal, “Effect of Velocity on the formation of rolled-up vortex on the delta wing using a water tunnel technique,” in *AIP Conference Proceedings*, American Institute of Physics Inc., Jul. 2020. doi: 10.1063/5.0013206.

[15] S. B. Wibowo, Sutrisno, T. A. Rohmat, S. Siswantoro, D. Fernando, and W. F. Naufal, “Effect of Velocity on the formation of rolled-up vortex on the delta wing using a water tunnel technique,” in *AIP Conference Proceedings*, American Institute of Physics Inc., Jul. 2020. doi: 10.1063/5.0013206.

[16] R. D. Firmansyah, S. Bekti Wibowo, and R. Mareta, “Aplikasi Pengukur 3 Derajat Kebebasan Gaya Aerodinamika Pada Water Tunnel,” *Jurnal Sains dan Teknologi*|, vol. 6, no. 2, 2017.