The recent progress and state-of-art applications of aerodynamics for airplanes

Yurou Wang *

Tianjin Nankai High School, Tianjin, China

* Corresponding Author Email: 18409455@masu.edu.cn

Abstract. In general, computational fluid dynamics (CFD) and wind tunnel experiments are the two core and widely implemented approaches to aircraft design and testing. This paper summarizes current developments in terms of CFD and wind tunnels in aircraft design and engineering. Information retrieval and Literature analysis are employed in order to investigate the state-of-art approaches and applications. Specifically, computational investigation of static height stability and aerodynamics of ground effect vehicles is discussed. According to the analysis, the ground effect is important for improving the realism of piloted simulation and estimating key crosswinds. The analytical modelling helps for better pilot training on various flight simulators. Moreover, further studies focus on increasing the realism of CFD simulation results ought to be carried out. These results offer a guideline for the investigation of the feasibility of CFD and wind tunnel experiments.

Keywords: Engineering, Aerodynamics of airplane, Simulation of CFD, Air tunnel.

1. Introduction

In retrospect, the aerodynamics is born and developed fastly after the construction of theory of air resistance in 1726 proposed by Newton. Subsequently, one of the milestones is appeared, i.e., the well-known publications of Hydrodynamica by Bernoulli, which gave a specific definition and derived a basic descriptions/link between the parameters (pressure, density, and flow velocity). Afterwards, the he also improved the conventional Euler equations to fulfill the conditions of Compressible and incompressible flow [1]. Later, the viscosity effects have been taken into consideration in the above equations, i.e., the Navier-Stokes equations, which are the governing equations of hydrodynamic. However, it is almost impossible to derive an analytical solution for the equations in most of configurations and initial boundary conditions. With the development of computer techniques, it is possible to solve the equation numerically in terms of the concepts of CFD. By excluding terms describing viscous actions, these equations can be simplified to produce the Euler equations. Additional simplifications, such as omitting terms representing vorticity, result in the complete potential equations.

During the Cold War, a series of high-performance aircraft were designed [2]. Computer simulations of flow qualities around complex objects have evolved to the point that entire airplanes may be constructed using computer simulations, with wind tunnel and flight tests to validate the computer predictions. Since the 1960s, aerodynamicists' priorities have evolved from studying fluid flow to designing a vehicle that interacts reliably with it. The war has triggered the development of the aerodynamics in pursuit of faster speed and better performance aircraft.

Previously, 2D analytical solutions for simple case (e.g., the linear potential) was given by plenty of scholars. Among them, the calculation of Richardson is one of the earliest examples that takes into consideration based on the concepts of CFD. Although they failed spectacularly, laid the groundwork for current computational fluid dynamics (CFD) and numerical meteorology. Indeed, similar approaches are also proposed in other researchers.

With this in mind, this paper will summarize the recent research of CFD simulation and wind tunnel test. for the sake of figuring out the latest application of aerodynamics of airplane. The reminder of the paper is separated as follows. The Sec. 2 will introduce the fundamental equations for hydrodynamic mechanics and milestones in the milestones in aircraft design and main components and functions of aircraft. The Sec. 3 will explain in detail about the definition of Computational Fluid

Dynamics (CFD) and state-of-art experiments. The Sec. 4 will describe in detail definition of wind tunnel and experiments including definition. Afterwards, the current limitations and drawbacks will be discussed and the future prospects will be proposed in Sec. 5. Eventually, a brief summary will be given in Sec. 6.

2. Basic Descriptions

2.1. Basic equations of fluid mechanics

Generally, the fluid mechanics cover three equations, i.e., the Buoyancy equations, Bernoulli's equation, Naiver-Stokes equation [3], which can be described as following respectively:

$$F_b = -\rho_f V_{\mathrm{imm}} g \tag{1}$$

$$p + \frac{\rho u^2}{2} + \rho g h = Constant \tag{2}$$

$$\rho\left(\frac{\partial u}{\partial t} + u \cdot \nabla u\right) = -\nabla p + \nabla \cdot T_D + f \tag{3}$$

Here, F_b represents for Buoyant force, ρ_f is the mass density, g is the acceleration of gravitational field, V_{imm} denotes for the Immersed volume of body in fluid, p is the pressure, u is the velocity, ρ denotes for the density, h is the height, T_d means the deviatoric stress tensor, f is the volume density of the body forces acting on the fluid.

2.2. Milestones in aircraft design

On December 17th, 1903, the first plane "Flyer 1" made by Wright Brothers was successfully tested in North Carolina, USA. Since then, gliders, propellers, jets, helicopters and civil aircraft have been invented one after another [4].

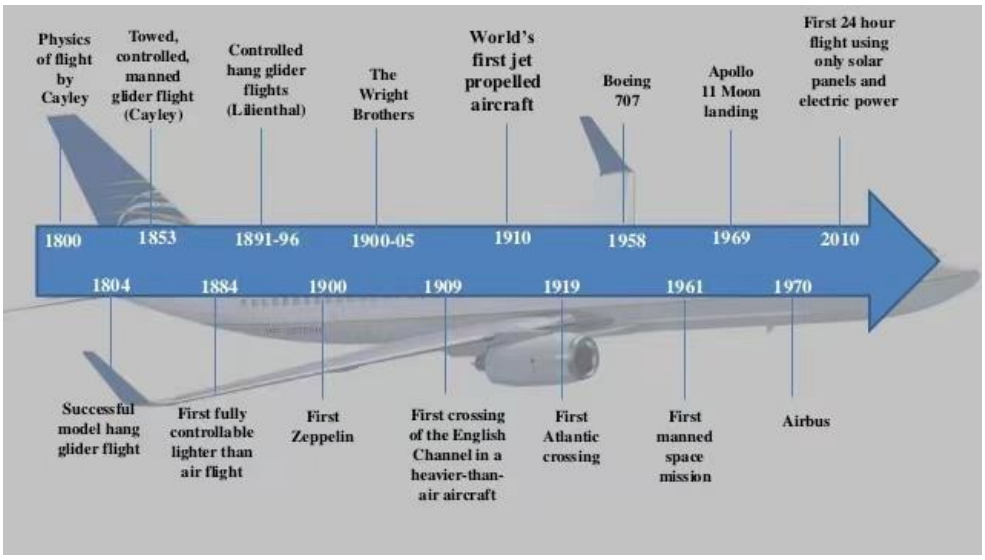

Figure 1. Timeline of Aviation [5].

According to Figure 1, Humans have long wished to fly like birds, including the ancients' usage of stones and spears, the ancient Greek Al Kutas' mechanical pigeons, and boomerangs as far away as Australia. Additionally, myths and tales portray human beings' longing to fly freely in the sky like birds. Human beings have acquired certain manual techniques over time, and the ancient Chinese invented kites, sky lanterns, and other flying things. After the Renaissance, European scholars began to investigate and experiment with scientific methods in order to design piloted aircraft. After the mid-twentieth century, humanity entered the realm of space exploration. Aviation has developed into the most advanced and influential branch of science, technology, and industry at the moment.

2.3. Main components and functions of aircraft

As shown in Fig. 2, the fuselage holds the plane together, with the pilots in front and passengers and cargo in the back. Wings provide lift, which is essential for flying, but several components of the wing govern the quantity and direction of lift. Ailerons are positioned on the trailing edge of the wing and help regulate the plane's roll. To turn left in the cockpit, lift on that side is reduced by the left aileron and increased by the right aileron. The plane begins to roll to the left and turn[7]. The tail of an aircraft is utilized for stability and lift when combined with the wings. The horizontal stabilizer maintains the plane's nose level (pitch). The elevator is a primary flight control located on the horizontal stabilizer. The vertical stabilizer allows the plane to weathervane. This prevents the aircraft's nose from swaying (yaw). The left and right pedals control the rudder, which is the principal flight control on the trailing edge of the vertical stabilizer. Pushing the left pedal causes rudder deflection to the left, causing the tail and nose to yaw left. Using the rudder and ailerons together generates synchronized turns [8].

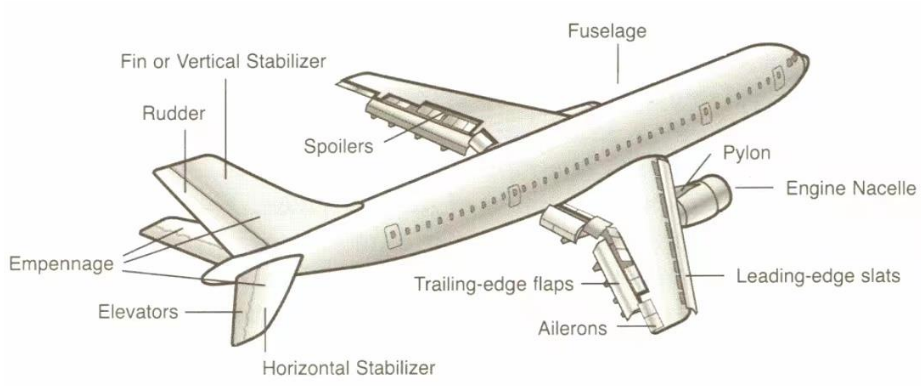

Figure 2. Main components of aircraft [6].

3. Computational Fluid Dynamics (CFD)

3.1. Definition

Computer-aided fluid dynamics (CFD) employs numerical solutions to the fluid equations for the sake of investigating the relationships between hydrodynamic variables [9]. Computers simulate the free-stream flow of fluids as well as their interactions in the boundary phases for different initial conditions. Better solutions are possible with high-speed supercomputers, which are often required for the most difficult issues. New software for sophisticated simulations (e.g., transonic or turbulent flows) is being developed and implemented to the state-of-art approaches. Initial software validation generally involves experimental equipment (e.g., the wind tunnels). Flight tests are commonly used for final confirmation. Research and technical difficulties involving fluid flows and heat transfer as well as visual effects for cinema and gaming are all solved with CFD.

3.2. Ground effect on vehicle static height stability and aerodynamics simulation

Compared to the airplane, the lift and drag enhancements are large, but the WIG effect vehicles still face the stability issue. On this basis, previous studies carried out CFD simulations and multi-tasks optimization to improve sectional shapes for the WIG and lift-to-drag ratio [10]. A flat lower surface and a convex lower surface near the trailing edge were shown to be excellent options in the optimization study. Three featured optima with high stability, performance, and relative stability are studied computationally for aerodynamics, stability, and three-dimensional effects. The Fig. 3 presents the simulations results accordingly [11]. According to the results, NACA4412, which is an airfoil that can be used for many different things, isn't as stable or aerodynamic as the other airfoils. For safety reasons, it is important to keep the WIG vehicle at the same height while it is running because it doesn't have enough stability when its height rises.

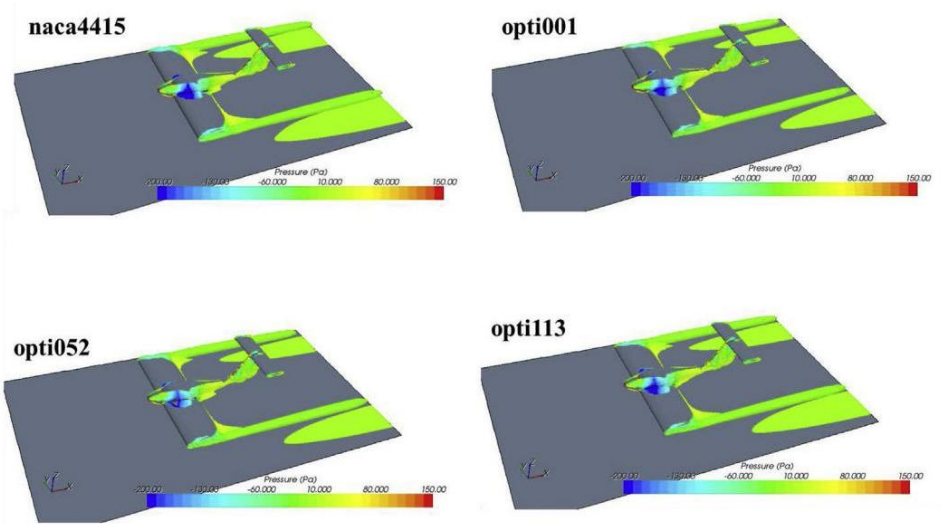

Figure 3. Iso-surface of wing tip vortex at h/C = 0.6 [12].

Pressure drops at the top of a wing, and the effect of a wing tip vortex and the fuselage is important. In ground effect, the path of the wing tip vortex is different from the path outside of ground effect, but it's the same in both places. There is a lot of pressure under the wings when they are in ground effect, so the vortices move away from the wings. The middle of the half-span is where the lower wing surface has the most pressure because the fuselage pushes air out to the side. Controllability will be better if the tail is above the height that this study looked at. With the suitable wing sections, a tail area that is about one-fourth the size of the main wing can keep the plane stable when it is in ground effect. Usually, WIG effect vehicles have a lot of space on their wings to keep them stable, which causes them to break down. More study of the flow interaction is very important [12].

3.3. Ground effect on aircraft lateral directional stability

CFD methods are used to run computer simulations of the aerodynamic characteristics of the CRM, which is a typical transport airliner. These simulations are done near the ground. Using the results, the stability and controllability of the plane's lateral-direction motion can be checked. This is because the aerodynamic forces and moments change with the bank angle. The most important changes in the lateral-direction modes near the ground have been found. This change in the lateral-direction dynamics in a piloted simulation could change how the plane responds to crosswinds, how it handles, and how realistic crosswind landings are. There was a meeting of aeronautics and space sciences in Europe called EUCASS-2017. This paper is based on that meeting. To figure out how ground effect aerodynamics work in a high-lift configuration based on hybrid models from DLR and NASA, more work is done. These models have flaps and slats fully deployed [13].

The “aerodynamic banking stiffness” was a new factor that was used in the eigenvalues analysis. The rolling and yawing moments of the plane changed depending on the plane’s bank angle. This "aerodynamic stiffness" has a big impact on how well an airplane can be controlled when it's close to the ground. Flight simulation results for aileron and rudder controls are shown in Ref. [13] for the 6-DOF flight. Changes in the plane's response to pilot control inputs happen when the plane is at a low, reduced altitude. It could change how the plane handles at low altitudes, which could have an impact on crosswind landings and the start of pilot-induced oscillations, among other things [13].

3.4. Investigation Recent development of a CAE-AVM-based CFD-wind tunnel correlation research

When the CAEDNW workshop on comparing computer-generated designs with those made in a wind tunnel is held, the experimental setting for wind tunnel will be demonstrated. To be specific, it will be going to talk about how the aero-elastic deformation of the wind tunnel model and the system that supports it can have an effect on how well the wind tunnel model works and how well CFD simulations work. Four study cases will be discussed, the geometry, and the mesh preparation of the workshop. A detailed summary of the CFD results from both the organizer and the people who took part is given. Major findings, both from CFD to CFD and from CFD to wind tunnel, are grouped together. It shows the importance of taking these things into account at a high subsonic Mach number of 0.85 [14].

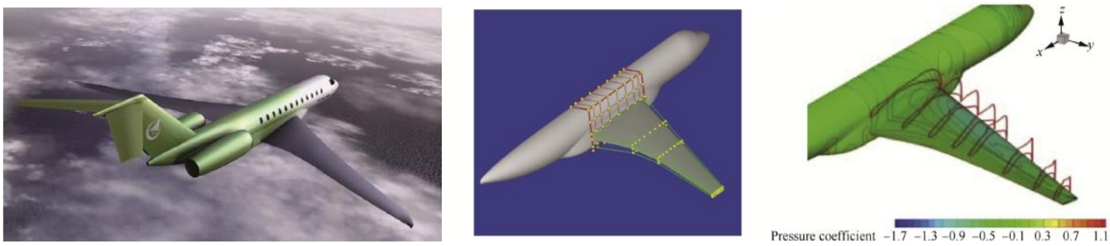

Figure 4. Wing optimization with FFD approach [14].

CAE's own code AVICFD-Y is utilized to simulate and evaluate the flow and figure out the performance of the airplane. A RANS solver based on multi-block meshes can do big flow analysis on multiple computer clusters, which is the reason that it is called multi-block mesh. A 13 percent thick supercritical airfoil with front loading is the basic shape of the wing. It reduces the pitching moment and improves the off-design performance of the wing, which is called NPU-SP6. Both the inverse method and numerical optimization are used to make the wing design better. Fig. 4 exhibits the FFD method for wing optimization. The maximum lift to drag ratio L/D at cruise speed is estimated by CFD to be 20 [14].

4. Wind Tunnel

4.1. Definition

Wind tunnels are enormous air-filled tubes used to simulate the interaction of air and an item in flight or on the ground [15]. The first wind tunnels were created in the late 1800s, when various researchers tried to design practical heavier-than-air flying aircraft, Contemporarily, wind tunnels are widely applied to study aircraft flight. Air and spaceship models are tested in wind tunnels. Some wind tunnels can accommodate full-size cars. A wind tunnel simulates flight by blowing air around an object [16].

4.2. Definition Inverted Joined Wing Wind Tunnel Tests and Comparison to CFD Results

Galinski et al. discusses and introduces the general assumptions and the process used to make a dual use inverted joined wing scaled demonstrator for wind tunnel tests. It also introduces about the aerodynamic properties that were found from these tests. During this part of the program, one is able to learn about the plane's exact properties. In the first place, the wind tunnel tests gave us a lot of information about the clean aircraft configuration's aerodynamics. This data was used to figure out how well the plane would do. In addition, large amounts of asymmetric initial settings simulations were run, i.e., the direction and lateral stability will be judged in terms of the aerodynamic features. When the plane sideslips, which happens a lot, it was very important to be able to predict how the plane would behave. This happens a lot, mostly when the plane is landing in a crosswind. Finally, a lot of trials with different settings of control surface deflections were carried out to check the overall performance. On this basis, the airplane's controllability was evaluated as a whole [17]. The data gathered is very important to the project's progress and results because it allows us to plan for the flight tests stage and to do the next steps, which are called airplane dynamic stability analyses, in the right way. As a result, compared to the computer results, one can make sure it is correct and draw conclusions for future research.

It is a closed-circuit wind tunnel with a 5m×6.5m open measurement section (as given in Figure 5). At that time, the measuring area's greatest air speed was 57 m/s, or 1990 N/m^2. The stream in the measurement region travels steadily at 0.5 percent longitudinal turbulence. A 7m 8-blade fan driven by a 1.5 MW (6 MW) electric motor creates the flow. This motor has been retired. The model was supposed to cruise at around 25m/s, thus the tunnel's air speed was set to this figure. Figure 6 also shows the I6B200 strain-gage aerodynamic balance placed inside the fuselage to assess model loads. The balance was connected to the HP 75000 data collecting system and the FFA-1211 amplifier. The balance's strain-gage bridge voltages were matched to the model's loads.

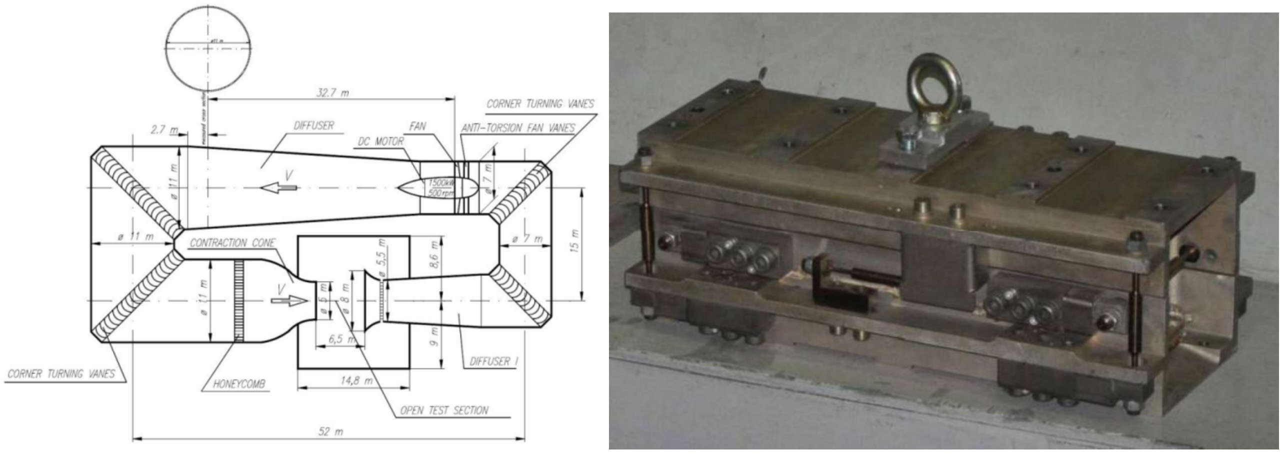

Figure 5. A sketch of Wind tunnel (left panel) and I6B200 aerodynamic balance (right panel) [17].

4.3. Wind-Tunnel and CFD Studies of Laminar Wing Transition and Performance

This paper builds on Hue et al.'s study of how airfoils alter and drag. The adoption of natural laminarflow technology in the actual world is a step closer once these investigations were extended to three-dimensional civil aircraft configurations. The goal of this research is to test a low-swept wing for business jets. Laminar/turbulent transition measurements with temperature-sensitive paint at Mach and Reynolds values typical for cruise flight [18]. Tollmien-Schlichting and crossflow transition criterion were utilized in the fluid dynamics calculations on this plane's shape. This research uses experiments and computer simulations to examine how well they predict change for three alternative approaches. When the plane is in cruise lift, extensive patches of laminar flow can be visible on the wing. This can minimize 10–15% of the plane's drag.

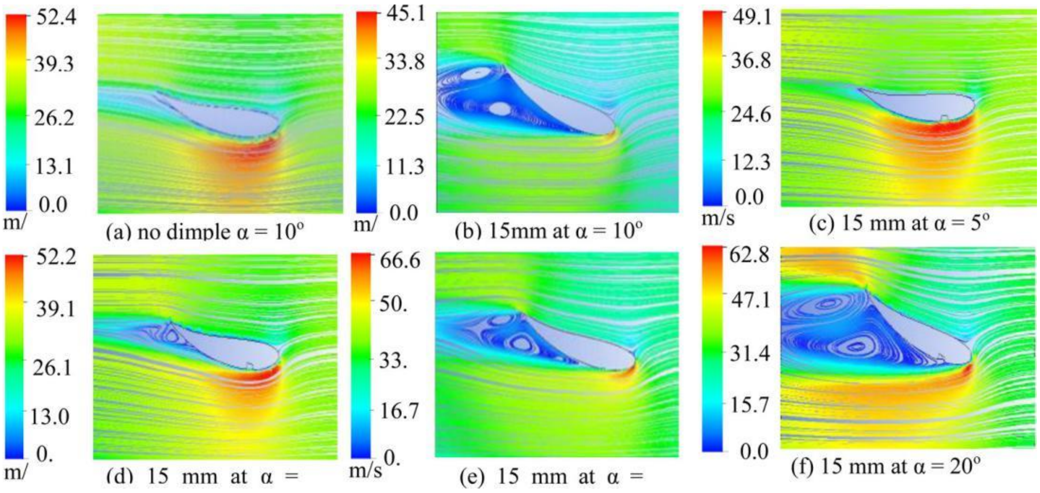

Figure 6. The velocity streamlines presented by CFD simulation [19].

4.4. Surface flow modification

In some cases, though, dimpling may not be very useful for racing cars with low AoA wings set up at the right speed [19]. Analyzing 2D CFD simulation data shows where the air flow separates on the aerofoil surface (seen from Fig. 6). A clear air flow whirling feature can be visible up to 20 degrees wing AoA after the dimpling point. It goes towards the trailing edge zone, where vortex shedding clearly separates the flow, and its intensity increases with wing AoA up to 20 degrees. The speed is higher on the top camber, and near the front edge. The dimples are put here to increase lift-to-drag ratio. Dimples on the upper surface of the aerofoil might produce flow separation earlier at the same angle of attack. Because a clean aerofoil with no dimples performing at the same angle of attack shows no flow separation. The flow separation point is delayed, and the lateral delay point shifts towards the trailing edge near the wake. To sum up, experimental result is in agreement with simulation results. The lift-to-drag ratio increases with aerofoil AoA, as predicted by CFD and tested in a wind tunnel.

5. Gaps and future outlook

Results from computational fluid dynamics simulations must be verified in a wind tunnel or in a real-world physical flow field before being considered reliable. As a result, the accuracy of CFD technology's computational results is frequently dependent on the consistency of the boundary conditions and physical parameters used in complex flow field simulations with reality, as well as the accuracy of the computational methods and post-processing, which are both important factors.

To starters, CFD can assist you save considerable amounts of time. Due to the computer's high processing power and massive storage capacity, CFD can create a virtual flow field model and run fluid dynamics simulations, significantly shortening the time required to design, construct, and test a wind tunnel experiment and avoiding the disadvantage of time-consuming physical structure change. Second, CFDs can lead to significant savings. CFD incurs no expenditures other than those connected with the acquisition, maintenance, and simulation of computer equipment. Since no physical facilities (e.g., the wind tunnel) are required, the cost of investing in CFD technology is extremely low. Most significantly, CFD can likewise ensure that operations are conducted safely. CFD technology is believed to be safer than wind tunnel trials since it does not expose people or items to a potentially dangerous environment.

6. Conclusions

In summary, this paper discusses the feasibility of CFD and wind tunnel from the perspective of aerodynamics of airplane. Based on the analysis, CFD technology has the potential to improve convenience. It is not only a necessary tool and concept in the field of fluid machinery research now and in the future, but it can also focus on providing designers with new ideas, allowing designers to achieve the greatest economic benefits in the shortest amount of time. CFD technology is a higher-performing, more robust, and more efficient design concept and scheme based on fluid flow mechanisms. Simultaneously, it can satisfy more constraint mechanism conditions and generate the optimal design scheme by combining constraint adjustment. The establishment of CFD as a method opens up new possibilities and approaches for designing fluid machinery. Because many programs are developed and researched on the basis of original research findings, their application scope is significantly limited. This article is based on recent research findings regarding the use of CFD in the design of fluid machinery, and how this technology constructs a general program for various speed ranges. According to previous studies, In recent years, scholars have applied computational fluid dynamics (CFD) numerical simulation technology to the study of wind-induced structural effects, a hot topic in the field of wind engineering. The selection of turbulence models is critical during the numerical simulation process. Due to the complexity of the flow around blunt bodies, existing turbulence models suffer from limitations, and no model can resolve all issues. Developing a model of turbulence with a high Reynolds number will be a difficult task. When the numerical convergence and divergence problems are applied to the flow around a blunt body, the problem becomes more serious. Convergence of six Reynolds stresses in the RSM Reynolds stress model is a critical scientific research problem; developing more stable and effective algorithms is also a critical scientific research problem.

However, some issues remain, such as the large amount of calculation required and the difficulty in achieving convergence. To address this issue effectively, we will need the collaboration of researchers from a variety of disciplines. Nevertheless, accuracy issues ought to be addressed. In the future, Numerous professional fields are actively innovating and developing as a result of the continuous advancement of science and economics. In the petrochemical and aviation industries, new fluid machinery design technology is critical to advancing the industry. As society advances, the demand for new technology in fluid machinery design increases. Traditional technology is no longer capable of meeting market demands and has been gradually phased out, necessitating the development of new technology with increased performance. After entering the new urbanization process, it is necessary to modernize the design management and concept. Designers must first master the skill set required in this field, which includes not only knowledge of historical design concepts, but also the ability to tailor a design scheme for this field that is compatible with the current modernization process. Then, designers must master the follow-up process for fluid machinery design after it is introduced to the market, while also meticulously controlling the internal flow state of the fluid machinery. The design scheme for fluid machinery is constantly being evaluated and combined with new technology in order to eliminate resistance and collisions and to reduce resistance and collisions. As a result, designers require computational fluid dynamics (CFD) technology to significantly improve the performance of fluid machinery.

CFD could save a large amount of time and money, while also ensuring operational safety. It also has a promising future in terms of development and space exploration. Overall, these results pave a path for aerodynamic simulation in aircraft design and shed light on guiding further study focusing on stability and safety of aircraft.

References

[1] H. Babinsky, How do wings work?. Physics Education, 38(6), (2003), 497.

[2] S. Eberhardt, Technology innovations in World War I airplane design. SAE International Journal of Aerospace, 8(2), (2015), 316.

[3] Information on: https://en.wikipedia.org/wiki/List_of_equations_in_fluid_mechanics

[4] J. D. Anderson, A history of aerodynamics: and its impact on flying machines (No. 8). Cambridge university press, 1998.

[5] Information on:https://www.bsea.nyc/6th-grade-timeline-project-history-aviation-due-1-18-18/

[6] Information on: https://airportparkingreservations.com/blog/how-to-overcome-fear-flying/plane-parts

[7] P. Jamieson, Innovation in wind turbine design. John Wiley & Sons, 2018.

[8] S. S. Rao, The finite element method in engineering. Butterworth-heinemann, 2017.

[9] C. Galinski, et al., Wind Tunnel Tests of the Inverted Joined Wing and a comparison with CFD results. Aircraft Engineering and Aerospace Technology, 2018.

[10] R. L. Bengelink, P. E. Rubbert, The Impact of CFD on the airplane design process: Today and tomorrow. SAE transactions, 2059-2068, 1991.

[11] S. A. Erturk, A. Dogan, Dynamic simulation and control of mass-actuated airplane. Journal of Guidance, Control, and Dynamics, 40(8), (2017), 1939-1953.

[12] J. Lee, Computational analysis of static height stability and aerodynamics of vehicles with a fuselage, wing and tail in ground effect. Ocean Engineering, 168, (2018), 12-22.

[13] M. Sereez, et al, Impact of ground effect on airplane lateral directional stability during take-off and landing, 2018, unpublished, information on: https://dora.dmu.ac.uk/handle/2086/16018.

[14] H. U. A. Jun, et al., Recent development of a CFD-wind tunnel correlation study based on CAE-AVM investigation. Chinese Journal of Aeronautics, 31(3), (2018), 419-428.

[15] J. B. Barlow, W. H. Rae, A. Pope, Low-speed wind tunnel testing. John wiley & sons, 1999.

[16] F. Campagnolo, et al., Wind tunnel testing of wake control strategies. In 2016 American Control Conference (ACC) (pp. 513-518). IEEE, (2016, July).

[17] C. Galinski, et al., Wind Tunnel Tests of the Inverted Joined Wing and a comparison with CFD results. Aircraft Engineering and Aerospace Technology, 2018.

[18] D. Hue, et al., Wind-tunnel and CFD investigations focused on transition and performance predictions of laminar wings. AIAA Journal, 56(1), (2018), 132-145.

[19] R. Allarton, et al., Surface flow modification of aerofoils for automotive racing car applications. International Journal of Modern Physics B, 34(14n16), (2020), 2040096.