Research Article

Modeling and Simulation of Wake Safety Interval for Paired Approach Based on CFD

Xin He,¹ Yilong Ma,² Hong Yang,³ and Yaqing Chen³

¹Air Traffic Management College, Civil Aviation Flight University of China, Guanghan 618307, China
²Hope College, Southwest Jiaotong University, Chengdu 610400, China
³Civil Aviation Flight Technology and Flight Safety Research Base, Civil Aviation Flight University of China, Guanghan 618307, China

Correspondence should be addressed to Yaqing Chen; chenyaqingmail@sina.com

Received 16 July 2021; Accepted 29 November 2021; Published 30 December 2021

1. Introduction

In recent years, the airline industry develops rapidly and plays an important role in the world economy. The increasing trend in civil aviation transportation has remained high for years, and airport capacity has begun to show signs of decline. Most aviation hub cities have plans to rebuild and expand airports, so some large hub airports began to build closely spaced parallel runways (CSPRs).

Regarding the operation of CSPRs, due to safety considerations, the wake interval standard for single-runway operation is still adopted in China for the time being, which has not stimulated the expansion advantage of CSPRs. However, the paired approach (PA) mode was proposed by American scholars. The PA concept is the one that leverages the real-time navigation and communication capabilities of the ADS-B equipage initiative to increase airport capacity by performing simultaneous dependent approaches to CSPRs [1, 2]. This operation mode greatly increases the capacity of CSPRs when successive aircrafts are approaching under instrument meteorological conditions, which greatly reduces the wake interval and avoids the wake influence caused by the lead aircraft, providing the operation of CSPRs with a new direction [3].

In 2000, the concept of “safety zone” was introduced by Landry and Pritchett through the research of the paired approach [4]. The safety zone, also called the protection zone, of the paired approach mode is shown in Figure 1. It is an area in which the paired aircrafts can maneuver operate, and separation assurance and wake avoidance are provided. The front boundary of safety zone called collision safety limit (CSL). The rear boundary of the safety zone called wake safety limit (WSL). The American Mitte Company proposed a 3° offset paired approach, which can better avoid the risk of wake encounters using computational fluid dynamics [5]. This research result has played a great role in the research of the subsequent paired approach mode, which has provided theoretical guidance for a simultaneous instrument
approach mode implemented at the airports such as Boston and San Francisco [6–8]. In 2001, the Advanced Aviation System Development Center of the company and the Industrial and System Engineering Campus of Georgia Tech University tested the initial procedures for the paired approach on a flight simulator, laying a foundation for the further definition of cockpit missions and the development of a cockpit separation management system [9].

NASA proposed to use the Monte Carlo method for the simulation calculation of the wake encounter risk caused by lead aircraft and proved the applicability of the method. The safety zone of the paired approach and the risk of collision during a wrong approach of the lead aircraft can also be simulated by using the Monte Carlo method [10–12]. In 2014, the Langley Research Center used the method of constructing a wake kinematics model to calculate the wake encounter risk for aircrafts that implemented the paired approach mode, which provided a reference for the quantitative analysis.

With the development of new technologies in civil aviation communications, navigation, surveillance equipment, and air traffic control, the operation of the paired approach mode has become more safe and efficient with the support of advanced equipment such as ADS-B surveillance equipment and the next-generation warning system ALAS. This section will discuss in detail the advantages and disadvantages of these methods.

2. Research Method of Wake

To determine the wake influence range of the paired approach aircraft, the motion characteristics of the wake vortex field of the paired aircraft must be analyzed and studied. At present, the research on the motion characteristics of aircraft wake vortex field mainly includes the following methods: wake feature detection test method, wake feature capture modeling method, and wake numerical simulation method. This section will discuss in detail the advantages and disadvantages of these methods.

2.1. The Test Method of Wake Feature Detection. The wake vortex has unique electromagnetic scattering characteristics, so it is easy to be effectively detected by sonar, radar, lidar, and other sensors [19]. Carrying out the wake feature detection test is of great value to the research and application of military and civil aviation.

Wind tunnel test is a conventional method to study the characteristics of the wake field of an aircraft. This method can measure the velocity distribution of the wake field in a certain space behind the aircraft model under different configurations. However, this method is subject to the influence of the test section scale and is mainly aimed at the detection of the characteristics of the wake field near the aircraft. The process of the wake generation and rolling up is shown in Figure 2 [20]. Measurement methods usually
include five-hole probe sensor measurement method [21], particle image velocimetry measurement method [22], or hot wire wind velocity method for measuring the wake field velocity over time. Since the 1970s, a large number of documents have recorded the velocity and vorticity fields of various configurations of typical transport aircraft measured by the wind tunnel test [23–31]. This method has the characteristics of good vortex field quality and high measurement accuracy, but it also has the following shortcomings: the wind tunnel structure is relatively complicated, the manufacturing cost is high, the experiment area is large, and it relies on high-precision sensors.

The water tunnel is a common test which uses water as the medium for dynamic experiments. The water tunnel test is more suitable for the experimental observation and research analysis of certain aerodynamic problems than the wind tunnel test, such as the generation and shedding of aircraft wingtip vortices [32]. The advantage of the water tunnel test is that the multiscale vortex structure near the wall, the wake vortex field shedding structure, and the spatial vortex separation structure can be clearly understood. The observed vortex field structure will not be interfered with by the tracer particle material properties, and the effect will be impaired [33, 34]. In addition, the water tunnel experiment also has the following shortcomings: the water tunnel structure is relatively complex, the manufacturing cost is high, the research period is long, and it is easy to cause experimental system errors due to the interference of the subsystems in the vortex field and the influence of the fluid medium or model. Especially most experiments use scaled models for experiments, and the simulated vortex environment cannot be absolutely consistent with the actual working conditions.

2.2. The Method of Wake Feature Capture Modeling. In order to further simplify the analysis process and enhance the prediction effect, scholars have improved the modeling method of the wake vortex field based on the observations obtained from flight tests. Since these wake velocity field modeling methods are based on the theoretical analysis methods of fluid mechanics, they are very effective for some uncomplicated vortex problems. However, for some more complex and nonlinear governing equations, they are beyond reach. Thus, it is suitable for qualitative analysis of simple vortex problems or complex fluid mechanics problems.

2.2.1. Lift Line Theory and Wake Vortex Hypothesis. From the classic Prandtl lift line theory, it can be known that if the wingspan of an airplane wing is constant, when it generates lift power, a pair of vortices in opposite directions will be derived behind it. The relevant variables of the above-mentioned reverse vortex pair can be calculated with the relevant performance data of the aircraft (maximum take-off weight, flight speed, wingspan, etc.). As a result, the calculation of wake development and evolution has been simplified. And this theory and hypothesis have been widely used in the study of wake analysis.

\[
b_0 = \frac{\pi}{4} b,
\]
\[
r_c = 0.05 b_0,
\]
\[
\Gamma_{co} = \frac{W_{T_0} g}{\rho U^2 b_0},
\]

where \(b_0\) is the distance between the left and right vortices; \(b\) is the wingspan length; \(r_c\) is the vortex core radius; \(\Gamma_{co}\) is the vortex ring volume; \(W_{T_0}\) is the maximum take-off weight; \(\rho\) is the air density; \(U\) is the acceleration of gravity; \(U\) is the velocity of the incoming vortex.

2.2.2. Convection-Diffusion Equation. The convection-diffusion equation is a nonlinear equation used to characterize convection and diffusion. And it is also a kinematic
equation, which is generally used in areas where the wake velocity is relatively stable, such as near and middle wakes. The mathematical expression when using the convection-diffusion equation for time-domain simulation is as follows [35]:

\[
\frac{\partial s}{\partial t} + u_i \frac{\partial s}{\partial x_i} - D \frac{\partial^2 s}{\partial x_i^2} = 0,
\]

where \( u_i \) is the rotational velocity field in the wake; \( D \) is the diffusion coefficient of air; \( S \) is the conservative passive quantity of the solution of the equation; \([y\bar{y}, 0] = y_0 - y\) is the initial condition.

2.3. Numerical Simulation of Aircraft Wake Based on CFD.

In order to make up for the difficulties such as high cost, long research time, and low accuracy of the test methods, the CFD method began to be applied to the analysis and prediction of the wake vortex field. In the 1960s, Takami used the vortex plate method for the numerical simulation of the wake, which can obtain the induced velocities at various locations in the vortex field [36]. Scholars visualized the time-based simulation and formed a wake model that can vary with a scale, which officially started the real three-dimensional wake simulation [37]. When the Reynolds average N-S equation appeared and began to be applied, CFD numerical methods began to be used to simulate the three-dimensional vortex field from the generation, development, and dissipation of wake [38]. Recent studies have shown a variety of numerical simulation methods, including large eddy simulation, which show good vortex characteristics for wake simulations, that is in good compliance with some phenomena observed in experiments [39].

In addition, the CFD numerical simulation method can also achieve faster simulation by combining classic wake models without the excessive pursuit of calculation accuracy. The initial wake vortex field distribution data obtained by the wake model can further develop the numerical evolution, which can predict the safety zone of the wake [39]. On the basis of retaining a certain calculation accuracy, the calculation efficiency is greatly improved, which is of great significance to the calculation and pre-diction of wake interval. The CFD method has unique advantages such as short calculation time, low cost, easy data extraction, and the continuous optimization and improvement of CFD technology, making it the main direction of fluid mechanics research.

3. CFD Modeling of Aircraft Wake Vortex Field

In order to analyze the motion characteristics of the wake vortex field of the paired aircrafts, the CFD method is used to construct the wake vortex field model of the paired aircrafts. The numerical simulation process of the wake vortex field is completed through preprocessing of the model calculation and setting of boundary conditions. Finally, by postprocessing the numerical calculation results, the wake vortex field of the paired aircrafts can be quantitatively analyzed.

3.1. Model Construction of Aircraft Wake Vortex Field

The construction of the aircraft wake vortex field model is an important part of the numerical simulation process. The density, quantity, and quality of the grid cells in the built model will directly affect the accuracy of the numerical simulation. The construction of the aircraft wake vortex field model mainly includes the following steps: establishment of the aircraft wing geometric model, construction and design of the aircraft wake vortex field calculation domain, and model calculation preprocessing (mesh division, dense calculation domain, setting boundary conditions).

3.1.1. Geometric Modeling of the Lead Aircraft Wing

In view of the extremely complex configuration of the aircraft wing, the entire wing is composed of several parts connected together. In addition to the main wing, there are multiple structures such as flaps, ailerons, spoilers, winglets, and engine pods. If the real wing structure is to be simulated numerically, it is not only difficult to model but also difficult to realize numerical simulation with the existing computer technology and CFD technology. Therefore, this paper simplifies the actual physical configuration of the wing. Based on the standard model of Boeing’s transport airliner, the modeling tool SolidWorks is used to intercept and construct the wing geometric model. According to the actual geometric size of the paired aircrafts, the model is scaled down to obtain a suitable geometrical model of the wing. Taking the B747-400 wing as an example, the wingspan of the B747-400 is 211 ft, and the reduced geometric model of the B747-400 wing is shown in Figure 3. During an aircraft approach, it usually flies at a certain angle of attack, and a certain angle of attack can be set when the model is built.

3.1.2. Calculation Domain Setting of Aircraft Wake Vortex Field

The computational domain of the vortex field refers to the mathematical operation (usually integral operation) during the numerical simulation calculation process. There are two ways to generate the computational domain—direct modeling and geometric extraction. In this paper, we need to study the wake vortex field of aircraft. The scale of the wake vortex is relatively large, so it is obviously impossible to generate the computational domain through geometric extraction. It is more appropriate to adopt direct modeling. In the actual calculation, in order to reduce the calculation load and the calculation pressure, the geometry of the calculation domain is usually simplified. Using the symmetry of the geometric model and the periodicity of fluid flow are common geometric simplification methods. For the numerical simulation of the external vortex field of the aircraft, the experimenter usually constructs the computational domain as a cylinder. In order to reduce the amount of calculation and take into account the motion characteristics of the wake, this paper simplified the calculation domain of
3.1.3. Calculation Model Preprocessing. The preprocessing process of the model generally includes meshing of the vortex field model, compacting the computational domain, and setting the boundary conditions. Compared to the construction and design of the computational domain, meshing is more important. The proper processing of the computational grid and the quality of grid generation are the primary conditions for numerical simulation calculations. The selection of grid quantity is generally based on empirical value or reference to the recommendations of relevant literature. In this paper, we use ANSYS ICEM CFD for preprocessing. It can provide advanced grid generation, geometry acquisition, and grid cell optimization functions for the analysis of complex models. It can also output grids for fluid mechanics solvers. The modeling tool SolidWorks will be used to intercept and build the wing geometric model. Then, it is imported into ANSYS ICEM CFD, the geometric model is checked, geometric repair is performed, and the above-mentioned calculation domain is generated. Then, the appropriate grid algorithm is selected to generate the computational grid of the geometric wing model and the computational domain, that is, the computational grid of the wake vortex field model. There are three grid algorithms provided by ANSYS ICEM CFD such as Octree algorithm, Delaunay algorithm, and Advance Front algorithm. The octree algorithm is commonly used for the formation of unstructured grid tetrahedron. This algorithm is more intelligent and automatic, so it was chosen for grid generation in this paper. In order to save computing resources and time, an unstructured grid is selected in this paper. The process of using ANSYS ICEM CFD for meshing is shown in Figure 5 in detail. First, whole grid cells are set, including grid parameters, plane grid, volume grid, prism layer, and period. Then, grid encryption is carried out for the computing domain by setting partial grid, plane grid, and line grid, and creating grid density region. Finally, meshing these grids, including three parts: meshing plane grid, meshing volume grid, and meshing prism layer grid. After the above steps, the wake vortex field of the paired aircrafts was meshed. The mesh model of the wing geometry model and the computational domain mesh model after the division are shown in Figure 6 and Figure 7, respectively. The X-axis indicates the direction of the incoming vortex, the Y-axis indicates the span direction, and the Z-axis indicates the height. The total grid quantity of the wake vortex field is about 14,000,000. Taking into account the principle of the wake generation during aircraft flight, the leading edge of the wing needs to be arranged with a dense grid so as not to affect the calculation accuracy. In order to capture the wingtip vortices, the computational domain grid of the area where the wake is formed downstream of the aircraft should also be for refinement processing. The grid division details of the leading edge of the wing are shown in Figure 8, and the details of computational domain grid refinement are shown in Figure 9.

For the finished grid, it cannot be directly used for calculation but must go through grid output setting and grid quality check. The grid quality is related to the calculation precision and also determines whether the calculation can converge. Check and Report Quality functions in ANSYS ICEM CFD can be used to check the grid. It mainly detects the geometric size, volume, and grid proportion of the mesh model, among which the minimum volume parameter needs to be paid more attention, and its value must be positive. When the grid model has a negative volume, it cannot be used for numerical simulation calculations. It is generally believed that the grid slope is better than 0.3. the Check and Report Quality functions are used in ANSYS ICEM CFD software to conduct a quality check on the grid model. The quality check result of the grid is shown in Figure 10. A small part of the grid has a slope of less than 0.3, and there is no negative volume grid. It is considered to meet the requirements of numerical simulation calculation.

4. Calculation of Wake Separation Based on CFD

There are two safe areas in the paired approach mode. They are the first safe area (the safety zone mentioned in Section 2) and the second safe area. This article focuses on the first safe area in the paired approach mode. The first safe area is the maneuvering flight area that is between the CSL and WSL of the paired aircrafts. The second safe area is an area where the wake of the lead aircraft has dissipated. Due to the
development of communication, navigation, and surveillance equipment, the CSL caused by the wrong approach of the paired lead aircraft can basically be ignored. Therefore, the research on the first safe area (safety zone) of the paired approach mode can focus on the WSL which needs to calculate the maximum wake safety interval.

The CFD-based wake safety interval calculation method combines knowledge and technology in the fields of fluid mechanics, mathematics, and computer science. The purpose of this method is to make up for the difficulty of the test method, the high cost, and the long research period. It can also make up for the shortcomings of using the wake model method to simulate complex wake vortex fields with insufficient accuracy. The calculation of the wake safety separation tends to be simplified under the premise of ensuring a certain calculation accuracy.

4.1. Determining the Initial Lateral Interval of the Paired Aircraft. For the 3° offset paired approach mode, before the trailing aircraft reaches the wake protection point, there is always a vertical safety gap of 1,000 ft between the two aircrafts, so the trailing aircraft does not need to consider the impact of the wake of the lead aircraft. When the trailing aircraft reaches the wake protection point, since the paired aircrafts do not need to maintain a vertical separation of...
1,000 ft at this time, the trailing aircraft must consider avoiding the influence of the wake of the lead aircraft. For the straight-in paired approach mode, since the distance between the runway centerlines of CSPRs is less than 2500 ft, a safe wake distance between the two aircraft should be determined at the beginning of the pairing to ensure flight safety. At this time, the initial lateral separation between the two aircraft is the distance between the centerline of the runway (C). Since the wake of the lead aircraft is generated near the wingtip, in order to make the calculation result more conservative, one half of the wingspan of the rear aircraft (B) should be deducted from the runway centerline spacing (D) as the starting side of the paired aircraft which is shown in Figure 11.

4.2. Numerical Simulation of the Lead Aircraft’s Wake Vortex Field. According to the methods and steps described above, the wake vortex field of the lead aircraft is numerically simulated. The result file is obtained after the numerical simulation calculation converges. The result file of the numerical simulation of the lead aircraft wake vortex field is postprocessed to obtain a visual lead aircraft wake vortex field (vorticity cloud map, vorticity isosurface map, etc.), as shown in Figure 12. Combining the basic process of the paired approach mode, the CFD-based paired approach wake safety interval calculation model is constructed.

Wing-tip vorticity, which is the wake of aircraft during flight, can be identified by vorticity. In TecPlot and CFD-Post, the variable vorticity needs to be customized. For example, in CFD-Post, new expressions and new variables can be created, and the two corresponding velocity differential variables can be loaded successively. The variable vorticity can be obtained by subtraction operation, and then the vorticity cloud map can be obtained. In TecPlot, the variable vorticity can also be obtained by creating a vorticity calculation formula in a similar way. The specific formula is as follows [24]:

\[ \begin{align*}
{\omega_x} &= ddy(\{u\}) - ddx(\{v\}), \\
{\omega_y} &= ddx(\{u\}) - ddz(\{v\}), \\
{\omega_z} &= ddx(\{v\}) - ddy(\{v\}).
\end{align*} \] (3)

4.3. Determining the Wake Safety Interval. As mentioned in the previous section, the strength and direction of the vortex can be described by the vorticity, so this article uses the vorticity as the physical quantity to identify the range of the wake vortex field. The paired approach mode requires the trailing aircraft to avoid the wake of the lead aircraft before the wake of the lead aircraft arrives so as to keep the trailing aircraft always maneuvering in the first safety zone. Therefore, according to the CFD-based paired approach wake safety interval calculation model, the maximum wake safety interval between the paired aircraft can be obtained. The wake safety interval (D) at this time is obtained from the vorticity distribution on the XY section (Y = D0) of the lead aircraft in the downwind direction. Assuming that the vorticity is on the XY section of the downwind direction of the paired aircraft, at the time Y = D0 and point X = D1, the absolute value is greater than 0, and the wake safety interval (D) of the paired aircraft can be obtained as follows:

\[ D = D_1. \] (4)

4.4. The Limitation of Present Work. This model can provide some theoretical reference for the operation of paired approach mode. However, the following aspects need further discussion and research: the wing model of the paired lead aircraft established in this paper is the standard wing model of transport aircraft. In future research, a specific model can be developed for each type of aircraft, to further reduce the error between numerical simulation test and real value. When constructing the calculation model of safe wake interval based on CFD, it is assumed that the distance between two aircrafts is the maximum safe wake interval when the absolute value of vorticity at the wing of the trailing aircraft is not zero, which is just to simplify the
calculation. In the future, the trailing aircraft can be assumed as a cuboid or ellipsoid to make it a more reasonable model.

5. Conclusion

In this paper, the computational fluid dynamics method is used to model the wake separation optimization when two aircrafts are operating in the paired approach mode. Through the theory introduction, modeling introduction in detail, and numerical simulation and analysis concluded that using the CFD method to study the safety zone of the paired approach wake is feasible. Comparing with experimental methods—wind tunnel test and water tunnel test, the CFD numerical simulation method shows great superiority. The research results of this paper can provide a certain theoretical reference for the study of the wake safe area of the paired approach mode in domestic air transportation.

This method also has some shortcomings that need to be improved. This method only considered the first safe area, ignoring the collision risk caused by the lead aircraft when it makes a wrong approach and intrudes in the trailing aircraft’s course. This would be the future research direction of the study. Also, a comparative study between experimental methods and CFD would be considered in the follow-up study.

Data Availability

The data used to support the findings of this study are available from the corresponding author upon request.

Conflicts of Interest

The authors declare that they have no conflicts of interest.

Acknowledgments

This research was supported by the Key Laboratory of Civil Aviation Flight Technology and Safety, CAAC Aviation Safety Capacity Building Fund Supported Project and CAAC Air Traffic Management Bureau Project.

References


[38] E. Coustols, L. Jacquin, and G. Schrauf, “Status of wake vortex alleviation in the frame work of European collaboration: