Aerospace Science and Technology 45 (2015) 60–66

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Analysis of jet blast impact of embarked aircraft on deck takeoff zone ☆

Yue Kuizhi [a,b], Cheng Liangliang [b,c], Liu Hu [b,*], Wang Yunliang [a]

[a] Department of Airborne Vehicle Engineering, Naval Aeronautical and Astronautical University, Yantai, 264001, China
[b] School of Aeronautic Science and Engineering, Beijing University of Aeronautics and Astronautics, Beijing 100191, China
[c] Department of Carrier-based Aircraft, Naval Aviation Institute, Huludao, 125001, China

ARTICLE INFO

Article history:
Received 24 December 2014
Received in revised form 7 April 2015
Accepted 18 April 2015
Available online 21 April 2015

Keywords:
Carrier-based aircraft
Aircraft carrier
CFD
Thermal coupling
Numerical simulation

ABSTRACT

The jet blast impact of embarked aircraft on the takeoff zone is studied to support carrier–aircraft adaptation. The geometric model of carrier–aircraft system is built in CATIA Software and an unstructured mesh of the model is generated in Workbench Software. The jet blast impact of embarked aircraft on flight deck and jet blast deflector while taking off from takeoff zone is numerically simulated with standard k–ε and three dimensional N–S equations based on CFD theory. Thermal coupling of wind over deck and jet blast is also taken into consideration. The distribution of temperature, pressure, velocity and streamline of the coupling flow is numerically simulated through Fluent Software and the temperature characteristics distribution on axes is also obtained. The results indicate: (1) the method presented in the paper can analyze fluid mechanics problems of complicated geometric models and obtain highly reliable simulation results; (2) the safe operation region for personnel, the suitable outfitting of jet blast deflector and proper layout of takeoff zone can be obtained.

0. Introduction

Aircraft carrier is an offshore strategic force projection platform of modern major naval powers; its main strike force—the carrier-based aircrafts can reach targets hundreds of miles, or even thousands of miles away by flight refueling technique. They can also execute important missions such as air defense, anti-ship strike, anti-submarine strike, shore strike, amphibious operation support and escorting mission. Aircraft carriers, despite their combat power, pose high risks. There are numerous equipment facilities placed on each area of the flight deck, including dozens of embarked aircrafts, motor tractors, fire-fighting trucks, cranes, etc. Fuel-serving facilities of aviation kerosene are placed on both sides of the deck, and there are dozens of tons of arms and ammunition carried by the planes and placed at the right side of the carrier. It is fair to say that working in the flight deck of a carrier is dangerous. The influence of high-temperature and high-pressure jet blast produced by the engine when an embarked aircraft is taking off on the aircrafts, personnel, vehicles and facilities on the deck is a key issue that needs to be addressed in the area of aircraft—carrier adaptation. Specific problems that need to be solved include the impact of jet blast (deflected after going through the bias panel) from an embarked aircraft in the takeoff zone on various areas of the flight deck, the influence of the coupling of wind over deck and jet blast on various areas of the flight deck, and what measures should be taken.

In order to study the carrier–aircraft adaptation problem of warming up of engines, the analytical methods of fluid mechanics are often used, and the common methods are empirical formula method of approximation, numerical simulation method, wind tunnel testing method and actual measurement method. The most accurate ones are wind tunnel testing method and actual measurement method. For the huge spending of them, the two methods cannot be rashly used from the very beginning of the study. Instead, the Computational Fluid Dynamics (CFD) method is generally adopted to numerically simulate the fluid characteristics of the aircraft [1]. The analysis of fluid characteristics of the aircraft with CFD method has been studied thoroughly both at home and abroad: in Ref. [2], the airflow over the deck is numerically simulated with a scaled model and the situation changes of the flow field under different working conditions are also studied. Ref. [3] uses CFD technology to study the airflow field of aircraft carrier and frigate, the results show that ship size, Island shape and position have influence on the vorticity and position of airflow behind the ship. In Ref. [4], based on CFD theory, the velocity field and pressure field of wind over deck and the ship bubbly wake are analyzed; in Ref. [5], the impact of engine wake flow on the deflector is numerically simulated with CFD theory, yet the model consisted of two cylinders and a deflector is simple, the impact of

☆ Foundation items: National Natural Science Foundation of China (51375490).
* Corresponding author.
E-mail address: 2674688972@qq.com (H. Liu).

http://dx.doi.org/10.1016/j.ast.2015.04.010

wind over deck on combustion-gas stream is not analyzed; Ref. [6] studies the corrosion of the jet blast deflector and the preventive measure; Ref. [7] studies the influence of noise from working engines of the embarked aircrafts on workers on the deck. Ref. [8] examines the management system of embarked carriers for dynamic scheduling in the flight deck and the hangar. The system takes into account the impact of the bias panel, the Island, the lift, the catapult, the arresting gear, etc. on the allocation and distribution of embarked aircrafts. The impact of jet blast on flight deck while multi-embarked aircrafts catapulting from flight deck is discussed in Ref. [9] based on CFD technology. Ref. [10] studies the vortex of all-wing aircraft based on CFD technology. The CFD method has been applied by Boeing in commercial plane design for more than 30 years. The progress of wind tunnel test, flight test and approximate methods of CFD used in the early days are introduced in Ref. [11]. Ref. [12] uses CFD method to evaluate the flight dynamics performance of the jet-propelled trainer and unmanned combat air vehicle (UCAV), etc. Ref. [13] uses unstructured mesh method to simulate the fluid mechanics property of aircraft with a small rocket booster under the fuselage. Ref. [14] examines the influence of the fairing of civil plane wings on lift-drag characteristics including lift, drag and moment. Research [15] has numerically simulated the aerodynamic characteristics of plane wings using CFD theory, compared and analyzed them with experiments, and found a good matching degree. Although many results have been achieved in the study of the key technologies in carriers and planes using CFD technology, research on the adaption of planes with carriers combined with CFD technology has not been satisfactory, as no published research report has been found about the adaption of planes with carriers combined with CFD technology, and no published academic paper has been seen about the influence of the thermal coupling of wind over deck and jet blast on carrier deck.

Using CFD technology, this research studies the influence of jet blast from embarked aircrafts on the carrier flight deck. It hopes to provide a direction for further research on carrier–aircraft adaptation, and offer references for the proper allocation of flight deck operations, so as to technologically support the research on the safety of workers on the carrier deck. The impact of embarked aircraft jet blast deflected by JBD (jet blast deflector) on flight deck is numerically simulated with CFD technology. The simulation method provides a solution for China and other aircraft carrier states of the world to study the impact of jet blast on flight deck. The solution can simplify and visualize some complex problems, and has a low cost and high accuracy.

1. Theory basis

The CFD simulation of the jet blast impact of embarked aircrafts on flight deck includes two theoretical bases: the turbulence model and governing equations.

There are popular types of turbulence models such as Spalart–Allmaras model, standard k–ε model, RNG k–ε model, Realizable k–ε model, standard k–ω model, etc.

Spalart–Allmaras model is a low cost turbulence model built for large mesh. It is suitable for simulating the internal and external flow with medium complexity, and the boundary-layer flow under pressure gradient; the standard k–ε model shows perfect robustness and is right for primary iteration, design selection and parameter discussion; RNG k–ε model fits solutions of fast strain, medium vortex and complex shear flow; Realizable k–ε model performs similarly to RNG k–ε, but has a better computational accuracy; Standard k–ω model is appropriate to be adopted to simulate near-wall boundary-layer separation, free-shear and low-Reynolds-number flow of fluid.

In the study of CFD simulation, the turbulence is numerically simulated through standard k–ε equations while the control theory adopts the three-dimensional N–S equations.

The standard k–ε equations are as follows [16]:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k \tag{1}$$

$$\frac{\partial(\rho \varepsilon)}{\partial t}+\frac{\partial(\rho \varepsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon \tag{2}$$

where ρ is the fluid density, k is the turbulent kinetic energy, t is time, u_i is the time averaged velocity, μ is the fluid kinetic viscosity, μ_t is the turbulent viscosity, σ_k is the Prandtl number corresponding to k, G_k is the production of k caused by mean velocity gradient. G_b is the production of k caused by the buoyancy. ε is the turbulent dissipation rate, Y_M is the contribution of pulsing stretching in the turbulence. S_k is the source item defined by the user. σ_ε is the Prandtl number corresponding to ε. $C_{1\varepsilon}$, $C_{2\varepsilon}$ and $C_{3\varepsilon}$ are empirical constants. S_k is the source item defined by the user.

The three-dimensional N–S equations of fluid mechanics control theory are actually momentum conservation equations, they are shown below:

$$\frac{\partial \rho u}{\partial t}+div(\rho u\vec{u})=div(\mu\, grad\, u)-\frac{\partial p}{\partial x}+S_u \tag{3}$$

$$\frac{\partial \rho v}{\partial t}+div(\rho v\vec{u})=div(\mu\, grad\, v)-\frac{\partial p}{\partial y}+S_v \tag{4}$$

$$\frac{\partial \rho w}{\partial t}+div(\rho w\vec{u})=div(\mu\, grad\, w)-\frac{\partial p}{\partial z}+S_w \tag{5}$$

where $\vec{u}$ is the velocity vector; u, v and w are the components on direction x, y and z respectively, p is the pressure on infinitesimal of fluid. $div()$ is the degree of divergence; $grad()$ is the gradient. S_u is the generalized source item of conservation of momentum equation on direction u. S_v is the generalized source item of conservation of momentum equation on direction v. S_w is the generalized source item of conservation of momentum equation on direction w.

2. Analysis of the influence of jet blast of embarked aircrafts on the carrier

The influence of jest blast (after deflection through the bias panel) of embarked aircrafts in the takeoff zone on the flight deck is numerically simulated using CFD theory. The simulation process consists of 4 steps—geometric modeling, mesh gridding, CFD solution and CFD analysis.

2.1. Geometric modeling

The conceptual carrier model and conceptual embarked aircraft model are designed with CATIA software with reference to the American Nimitz class nuclear powered aircraft carrier, shown in Fig. 1.

The conceptual aircraft carrier has one inclined and straight two-section flight deck, a carrier land, four lifts, three arresting gears, four catapults and one hangar. The flight deck is 337 m long and 77 m wide; the carrier is 76.8 m high.

The conceptual carrier-based aircraft is a heavy aircraft with one single seat, two engines and three airfoils. The aircraft has a nose-wheel landing gear, outward-inclining vertical double-fin, an internal weapon cabin, a DSI air inlet, S-shaped air suction, and a