Schemed Power-augmented Flow for Wing-in-ground Effect Craft in Cruise

YANG Wei, YANG Zhigang*
Shanghai Automotive Wind Tunnel Center, Tongji University, Shanghai 201804, China

Received 9 March 2010; revised 17 August 2010; accepted 2 December 2010

Abstract

To provide detailed insight into schemed power-augmented flow for wing-in-ground effect (WIG) craft in view of the concept of cruising with power assistance, this paper presents a numerical study. The engine installed before the wing for power-augmented flow is replaced by a simplified engine model in the simulations, and is considered to be equipped with a thrust vector nozzle. Flow features with different deflected nozzle angles are studied. Comparisons are made on aerodynamics to evaluate performance of power-augmented ram (PAR) modes in cruise. Considerable schemes of power-augmented flow in cruise are described. The air blown from the PAR engine accelerates the flow around wing and a high-speed attached flow near the trailing edge is recorded for certain deflected nozzle angles. This effect takes place and therefore the separation is prevented not only at the trailing edge but also on the whole upper side. The realization of suction varies with PAR modes. It is also found that scheme of blowing air under the wing for PAR engine is aerodynamically not efficient in cruise. The power-augmented flow is extremely complicated. The numerical results give clear depiction of the flow. Optimal scheme of power-augmented flow with respect to the craft in cruise depends on the specific engines and the flight regimes.

Keywords: ground effect; power-augmented ram; aerodynamics; engine models; numerical simulations

1. Introduction

The von Karman-Gabriell’s diagram[1] depicts the efficiency of various transportations, and introduces a delta zone near the center of the technology line presenting high-speed and excellent transport efficiency. None of the conventional means of transportation can fill this zone. As a promising means of transportation, wing-in-ground effect (WIG) craft, flying over the ground or water with the speed of 100-800 km/h and high lift-to-drag ratio, just fills the gap. One of the most important impediments to the development of WIG craft is the large power required for takeoff and landing. Many powered-lift schemes have been devised and tested to this end[2-5]. A potential solution to increase the landing and takeoff performance was found in the early 1970’s. In 1962, Ekranoplan of S M-2 was built in Russia[6], with a low aspect ratio wing and a large T-tail. Another unique feature is the jets that blew under the wing enhancing the air cushion, which was applied to most following WIG craft and is called power-augmented ram (PAR) engine. Some WIG craft only use PAR system during takeoff; notable examples of this are the KM and Lun Ekranoplanes that situate their jet engines forward of the wing and deflect the thrust downwards under the wing. Once in cruise flight, the engines for PAR could be throttled back extensively or some of them even shut off. Lippisch[7] employed PAR system on his X-112 craft and got an increase of lift coefficient by 25%. Consequently the takeoff weight can be raised and the takeoff distance, especially from water, can be shortened sig-
significantly. Since the PAR provides a substantial system performance improvement to aircraft designs flying in and out of ground effect, the potential advantage does not allow us to reject consideration of PAR in cruise. Design concept that cruises on PAR is economical as well. The additional drag caused by the sea’s rough conditions can be sufficiently reduced by the PAR effect. Related to the WIG drag in calm water, the additional drag is reduced from 42% to 24% due to PAR\[8\].

Different schemes of exhaust blown from the PAR engines onto the wing may be adopted, when modeling the aerodynamics of WIG craft in regimes of takeoff and landing by using PAR lifting system. Some of these schemes were depicted by Rozhdestvensky in his publication\[9\]. The interaction of exhaust from PAR engine with the main wing complicates the flow and affects the aerodynamics of craft. The current understanding, based on assumption and limited experiment, falls short of the detailed understanding of the flow. Computational simulations of these complicated flow fields become increasingly important in offering a means to study the flow and to minimize costly wind tunnel tests. Much work on ground effect of automotives and craft has been carried out by using the method of computational fluid dynamics (CFD) successfully\[10-15\]. Hirata, et al. conducted a numerical investigation on PAR wing in ground effect, considering two boundary conditions: 1) a moving belt ground plate condition; and 2) a fixed ground plate condition corresponding to a wind tunnel test[16]. Based on questions raised in applying PAR system in cruise, further study needs to be carried out. The present work focuses on investigating the flow characteristics of WIG craft with PAR system in cruise. A computation of the complex flow is presented by employing the CFD method to the study. Different schemes for the PAR system are included in discussion. One goal of the current study is to reveal the complex flow when considering the PAR scheme for WIG craft in cruise, as aerodynamic simulation details the flow. Additionally we wish to evaluate the schemes of the power-augmented flow in order to perform well in cruise. The study will hopefully clarify the understanding of the flow and resolve some of the uncertainties arising from previous studies.

2. Numerical Approach

The WIG craft model of the present work is shown in Fig.1. It has the similar size and weight as Boeing777, and is supposed to carry more payloads. A turboprop engine, mounted at the intersection of the vertical stabilizer and the tail plane, is omitted. A half-model is applied to numerical simulation, owing to symmetry of the geometry. The pitch angle of the craft is zero. The flight height \( h \) is defined by the distance from the ground to the lowest point of the WIG craft model, and is nondimensionalized by using the mean chord length \( c \). The simulation is conducted at a Reynolds number of \( 1 \times 10^8 \), based on \( c \). This corresponds to a Mach number of 0.5 for freestream velocity.

Fig.1  WIG craft model.

Fig.2 shows the computational grids. The structured-unstructured hybrid grid method is applied to the simulation. Hybrid grid methods are designed to take advantage of the positive aspects of both structured and unstructured grids. Structured grid is specified in the bulk of the domain, and unstructured grid in local regions for complex craft geometry. For near wall flow, a wall normal spacing of \( 2.0 \times 10^{-4}c \) is used for both craft and ground; the \( y^+ \) value on wing is about 100 and an average \( y^+ \) value of 200 for craft is achieved, which locates in log layer and is suitable for the non-equilibrium wall functions applied to current numerical simulation. Ten layers of cells are clustered towards the walls with a growth ratio of 1.2 in the wall normal direction. The total number of cells is approximately 3.0 million. This is thought by the authors to be rough to give accurate results but proper to capture the characteristics of PAR flow.

Fig.2  Computational grids.
CFD techniques for turbomachinery make it possible to perform CFD computations throughout the engine of aircraft[17]. These computations are very time-consuming. In most aerodynamic studies of aircraft with engines, such as study of DLR-F6 in AIAA Drag Workshops, the engines are usually modeled as hollow pipes, called “flow through nacelles”[18]. However it is unacceptable for the present study, as far as the characteristics of PAR engine is considered. For simplification, the PAR engine in numerical simulation of the present study is replaced by an outflow and an inflow boundary condition in the CFD model (see Fig.3). The numerical outflow boundary is the inlet of the engine. The numerical inflow boundary is the outlet of the engine.

![Fig.3  Simplified engine model.](image)

The use of CFD codes to simulate the flow around geometrically complicated shapes such as airplanes, cars and ships has become standard engineering practice in the last few years. A number of commercially available codes can be used to perform these studies. The finite volume code FLUENT is employed in the present study. It has been performing well in aerodynamic prediction for craft[19]. The finite volume code FLUENT is employed in the present study. It has been performing well in aerodynamic prediction for craft[19].

The governing equations are the compressible Reynolds-average Navier-Stokes (RANS) equations for continuity and momentum:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} = 0 \quad (1)
\]

\[
\frac{\partial (\rho u_j)}{\partial t} + \frac{\partial (\rho u_j u_i)}{\partial x_i} = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u}{\partial x_j} \right) \right] + \frac{\partial}{\partial x_j} \left( \frac{\partial (\rho u_i'u_j)}{\partial x_j} \right) \quad (2)
\]

where \(\rho\) is the air density, \(p\) the air pressure, \(\mu\) the dynamic viscosity, \(\delta_{ij}\) the Kronecker delta symbol; \(u_i, u_j\) and \(u_l\) are the velocities in the \(i\)th, \(j\)th and \(l\)th directions; \(-\rho u_i'u_j\) is the Reynolds stress term, and the realizable \(\kappa-\varepsilon\) turbulence model is used[20]. The transport equations of \(\kappa\) and \(\varepsilon\) are written as

\[
\frac{\partial}{\partial t}(\rho \kappa) + \frac{\partial (\rho \kappa u_j)}{\partial x_j} =
\]

\[
\frac{\partial}{\partial x_j} \left[ \left( \frac{\mu + \mu_t}{\sigma_\kappa} \right) \frac{\partial \kappa}{\partial x_j} \right] + G_{\kappa} - \rho C_k \varepsilon - Y_M + S_{\kappa} \quad (3)
\]

\[
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial (\rho \varepsilon u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu + \mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] +
\]

\[
\rho C_{1_s} \varepsilon^2 - \rho C_{1_v} \frac{\varepsilon^2}{\kappa + \sqrt{\varepsilon}} + C_{1_s} \varepsilon \frac{C_{3_s} G_s}{\kappa} + S_{\varepsilon} \quad (4)
\]

where \(G_k\) represents the generation of turbulence kinetic energy due to the main velocity gradients, \(G_k\) the generation of turbulence kinetic energy due to buoyancy, \(Y_M\) the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate, \(\mu_t\) the dynamic viscosity of turbulence, \(\nu\) the kinematic viscosity, \(S_k\) and \(S_{\varepsilon}\) are userdefined source terms, \(\sigma_\kappa\) and \(\sigma_\varepsilon\) the turbulent Prandtl numbers for \(\kappa\) and \(\varepsilon\), \(C_1\), \(C_2\), \(C_1\), and \(C_3\) are constants.

The realizable \(\kappa-\varepsilon\) model is a more advanced version of two-equation turbulence model. This turbulence model has been extensively validated and well behaved for a wide range of flows, including rotating homogeneous shear flows, free flows including jets and mixing layers, channel and boundary layer flows, and separated flows. It is chosen because this turbulence model has also been shown to prevent the turbulence energy from becoming negative and is more suitable for predicting the separation point. Most of these features are prominent in the flow of craft and aero engine. A second-order accurate scheme is used for the convective and viscous terms.

For the simplified engine model, the pressure needs to be defined at the inlet of the engine, while at the outlet the values for pressure and temperature are required. The ratio of mass flow rate for inlet and outlet of engine is controlled nearly by 1:1. For the far boundaries, pressure inlet boundary conditions is chosen for the inlet and pressure outlet boundary condition for the outlet; a no-slip boundary condition is specified to the craft and the ground, and the ground is considered to be rigid and has the same velocity as inlet; a slip boundary condition is specified to other boundaries.

3. Results and Discussion

3.1. Validation of computational method

Before numerically investigating the complex PAR flow, the present computational method is firstly validated on DLR-F6[21] wing-body configuration at \(Ma=0.75\) and Reynolds number of \(3.0\times10^6\). A wall normal spacing of 0.3 mm is used for grid points closest to the wall, which also yields 10 cells in the boundary layer region. The average y’ value is about 100. The total number of cells is 3.0 million which is equivalent to the coarse grid released by AIAA Drag.
Prediction Workshop. The drag polar is shown in Fig.4. Numerical results compare well with the experimental values, although the drag is somewhat higher, mainly due to fully turbulent computation. Furthermore, the turbulence model introduces more viscous effect.

Surface pressure profiles are compared in Fig.5, where $b$ is span. Computation predicts the shock location to be slight forward, and also exhibits a small difference at the maximum negative pressure. Grid refinement and an advanced turbulence model can improve the results. Taking into account the issue of PAR and ground effect, the positive agreement between numerical results and experimental values gives reasonable evidence that the computational method is a reliable choice for the current study.

### 3.2. Flow features

Simulations of WIG craft in cruise are performed over a wide range of deflected nozzle angles ($\theta_{\text{def}}$) from $0^\circ$ to $25^\circ$ with respect to level ground. Three typical PAR flows are captured and shown in Figs.6-7.

Most exhaust flows over the suction surface with small deflected nozzle angles. With certain deflected angle, a phenomenon of attaching to the surface for the jets, called Coanda effect can be seen. The high-speed exhaust flows along the upper surface from leading edge to trailing edge with attachment, and more suc-
Fig. 7 Total pressure distributions of different PAR schemes ($h/c = 0.1$).

Lift happens when negative and positive pressure respect to the reference pressure occurs on upper and lower surface respectively. Fig. 8 presents the static pressure distributions.
pressure distributions on upper and lower surfaces for \( h/c = 0.1 \). Velocity distributions in Fig.6 provide an overview of PAR flow around the wing. The disparity under the wing for static pressure distributions is illustrated here clearly. The PAR flow applied to the present work only increases the suction effect regionally. At the same time, very small or big deflected nozzle angle cannot make the wing hold high positive pressure.

There is pressure loss under wing at the leading edge and trailing edge besides wingtip. Acceleration for oncoming flow along the lower wing surface exists before air cushion forms. The acceleration will be promoted when nozzle angle increases, so is leading edge pressure loss. The trailing edge pressure loss is attributed to complex flow under wing that airflow accelerates to meet high-speed air over wing at trailing edge for very small nozzle angle; when enveloping the leading edge, jets leads more oncoming air flowing under wing and flows based on Coanda effect improve pressure distribution; as most jets are ducted under wing and flow featured by Coanda effect cannot be developed, momentum increase under wing results in serious pressure loss along the trailing edge.

Fig.9 details the pressure coefficient distributions for different deflected nozzle angles along the chord, in which \( c(z) \) is the local chord. The illustration shows a sharp positive pressure peak at the leading edge. It is proposed that additional energy is injected into the flow by jets. The value of this peak is high when the leading edge is enveloped. The suction effect induced by jets is mainly realized at the front part of the wing. And, the more jets flow over the wing, the more suction is realized. Reduction of pressure effect happens near the leading edge, with increasing deflected nozzle angle.

![Fig.8 Static pressure distributions of different PAR schemes (h/c=0.1).](image)

![Fig.9 Pressure coefficient distribution (2z/b=0.29).](image)

### 3.4. Aerodynamic characteristics

The PAR scheme for jets ducted under the wing is completely plausible for takeoff and landing regimes, which represent low speed and big angle of attack. Furthermore, the flap deflection angles at takeoff are quite considerable with small ground clearance. The deceleration of the jets in the channel under the wing can be observed, so is the remarkable ram effect. Strengthening ram effect is the best way to improve performance of takeoff and landing. While in cruise, it is not the case.

The aerodynamic performance with different deflected nozzle angles in cruise is shown in Fig.10. The thrust \( T \) is believed to be generated by the acceleration of the mass flow inside the engine. The lift and drag coefficients are defined as

\[
C_L = \frac{L}{\frac{1}{2} \rho v_u^2 s} \quad (5)
\]

\[
C_D = \frac{\Delta D}{\frac{1}{2} \rho v_u^2 s} \quad (6)
\]

where \( L \) is the measured lift, and \( s \) the area of the wing. The drag coefficient is based on the measured drag \( D \) minus the components of thrust in flow direction \( T_u \), that is \( \Delta D = D - T_u \). According to the definition,
the drag coefficient will decrease with increasing thrust, and a balance state of thrust and drag-self-propelling condition, \( C_D = 0 \), seems to be accessible\(^{[22]}\). Propeller engines have been utilized on most small WIG craft for PAR effect, such as the Volga-2 in Russia and the XTW in China, providing enough thrust to meet the total drag. Whereas, it is not practical for big WIG craft, for the PAR engines cannot provide the needed power alone. Therefore, turboprop engine is indispensable for big WIG craft, such as WIG craft model of the present work.

The aerodynamic lift due to forward motion in ground effect becomes significant in cruise with high-speed. The flow deceleration and ramming effect under the wing are weak after takeoff. Thereby, PAR scheme must be rearranged in order to achieve good aerodynamic performance when concept of cruising with combined power support is presented. The lift and drag increase with augment of the deflected nozzle angle firstly. This attributes to the realization of suction effect induced by the jets and the decreased components of thrust by which part of drag is overcome. Then the lift will decrease and the drag keeps nearly constant. It seems that realization of suction effect due to jets cannot be guaranteed with big deflected nozzle angles. Moreover, flow under wing is accelerated and pressure effect is weakened.

4. Conclusions

A computational study of the power-augmented flow for WIG craft in cruise is carried out. Typical schemes of PAR are captured and presented in the current work. The numerical results give detailed and general description of the flow structure when taking power assistance from PAR engine into account in cruise.

The discussion outlines some characteristics including distributions of pressure and velocity. Comparison shows that PAR scheme used to be favorable in takeoff and landing is not efficient any more and should be rearranged in cruise. Jet flow based on Coanda effect is an appropriate option, the suction effect over wing is strengthened, the pressure loss at the trailing edge is recovered to some extent and flow separation at the trailing edge can be delayed.

The schemed PAR flow is extremely complex. It features the interaction of air blown from the PAR engine with the wing. Unfortunately, only one pair of simplified engine models is employed to the current study. More in-depth and detailed research work is needed. Optimal performance of PAR effect in takeoff and landing depends on the specific scheme and on the specific design of the PAR engine. This is also clearly reflected in cruise.

References


Biographies:

**YANG Wei**  Born in 1980, he received M.S. degree in fluid mechanics from Nanjing University of Aeronautics and Astronautics and then became a Ph.D. candidate in College of Automotive Engineering, Tongji University. His main research focuses on fields of aerodynamics of craft and vehicles, and ground effect.

E-mail: david_yangwei@yahoo.cn

**YANG Zhigang**  Born in 1961, he is the managing director of Shanghai Automotive Wind Tunnel Center. He holds a Changjiang Professorship from Ministry of Education at college of Automotive Engineering, Tongji University. His main research interests are aerodynamics, flow instability, turbulence modeling and CFD.

E-mail: zhigangyang@tongji.edu.cn