

Numerical Simulation on the Ventilation Cooling Performance of the Engine Nacelle under Hover and Forward Flight Conditions

2011-01-0513

Published
04/12/2011Yongqi Xie
BeiHang UniversityHongxia Gao , Jianzu Yu and Jing Xiao
Beihang University

Copyright © 2011 SAE International

doi:[10.4271/2011-01-0513](https://doi.org/10.4271/2011-01-0513)

ABSTRACT

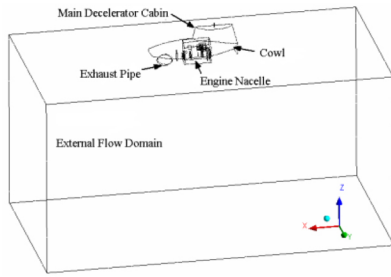
The main objective of this work is to investigate, by means of numerical simulations, the performance of the engine nacelle ventilation cooling system of a helicopter under hover and forward flight conditions, and to propose a simplified method of evaluating the performance based on rotor downwash flow by taking the synthetical effect of engine nacelle, exhaust ejector and external flow of a helicopter into account. For the engine nacelle of a helicopter, an integrated model of the nacelle and exhaust ejector was set up including the domain of external flow. The unstructured grid and finite volume method were applied for domains and control equations discreteness, and the standard $k-\epsilon$ model was applied for solving turbulent control equations. Using the business CFD software, the flow field and the temperature field in the nacelle were calculated for single inlet scheme and double inlets scheme, total up to 9 schemes. The performance of the exhaust ejector was computed. And the influence on the ventilation cooling performance of the nacelle was analyzed for different inlet number, inlet size and inlet position. The comparisons were completed between flight trials and numerical simulation results, and they show good agreement. The results show that the method can predict the performance of the ventilation cooling system of the nacelle. Moreover, the study indicates that, for the same inlet scheme, the temperature of the nacelle in forward flight is lower than that under hovering condition, but the pumping coefficient of the exhaust ejector in forward flight is higher. The inlet position should avoid being located on the underside of the cowl. It is

also indicated that the simulation results can be regarded as a reference for the design and optimization of the system.

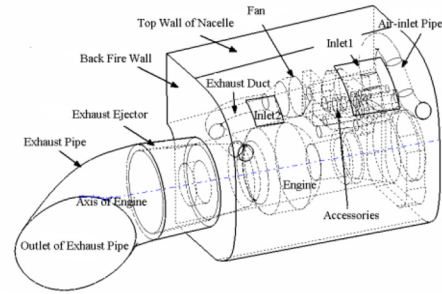
INTRODUCTION

In order to pump ambient air into an engine nacelle to remove the amount of heat dissipated by the engine and accessories, the exhaust ejector was generally used as the power supply to cool the nacelle. Under flight condition, it has important influence of rotor downwash flow and airstream disturbed by the fuselage configuration on the cooling performance of the nacelle.

In the field of rotor downwash flow, rotor/fuselage disturbing and exhaust ejector performance, many investigations have been conducted [1, 2, 3, 4, 5, 6, 7, 8, 9]. For example, Cao et. al. [1] used finite volume method to calculate the flow field characteristics around the engine inlet for MI-171V5 at low speed and hovering flight, the results showed that the CFD method can predict the vortex wake flow field distributions. Numerical study on a cylindrical shroud ejector and its integration with a subsonic mixed flow turbofan installation was carried out by Khalid and Sokhey [2]. Gupta [3] studied the performance of a simplified Quad Tiltrotor in forward flight using CFD, and analyzed the effect of the propeller downwash by simulating forward flight for the lowest advance ratio without the propellers and with the propellers rotating. Balland et. al. [4] summarized the joint study conducted by Airbus France and Snecma Moteurs to build up a methodology for aero-thermal studies of the engine nacelle. The impact of equipment items on flow behavior was studied and the weight of both radiative and convective transfers was



(a) Model including external flow domain



(b) Sketch of inner structure of the engine nacelle

Fig.1. Illustration of nacelle-ejector integrated model including the external flow domain

estimated. However, because it still has many problems directly using Navier-Stokes equation to simultaneously calculate the flow field of the nacelle, the rotor flow field and the interference of rotor/fuselage [10, 11], for thermal analysis of the nacelle, the effect of rotor downwash flow was usually taken into account by means of assuming the downwash velocity as a constant value, and also the affection of the exhaust ejector was taken into the count by assuming a proper boundary condition [6, 7]. But the methods above often neglect the effect of the fuselage on the nacelle inflow and can't predict accurately the air flow and temperature distribution in the nacelle.

Considering the problems of the present research, this paper proposes a simplified method of evaluating the ventilation cooling performance of the nacelle based on the downwash flow. In the method, the rotor downwash flow, nacelle and exhaust ejector will be considered together, and the integrated model of the nacelle and exhaust ejector was set up including external flow domain. For the engine nacelle of WZ helicopter, the performance of the ventilation cooling system was simulated in hovering and forward flight utilizing the business CFD software. The influence of inlet number (single inlet and double inlets), inlet size and inlet position was analyzed on the flow field and temperature field in the nacelle and the performance of exhaust ejector. The research has practical guidance significance for the design and optimization of the nacelle cooling system.

METHOD DESCRIPTION

PHYSICAL MODEL

Fig.1(a) shows an illustration of the integrated model of the nacelle and exhaust ejector including the external flow domain for the helicopter. The coordinate system of the model was the same as that of the helicopter. The middle plane of the helicopter was regarded as the symmetry plane while establishing simulation model. The domain size was 14m×8m×7m, and the distance from its upper surface to the top surface of the cowl was 0.5m. The shaft of the rotor wing was located at the center of upper surface of the main

decelerator cabin, and the distance from the outside apoapsis of the cowl to the shaft was 0.88m. The engine nacelle and exhaust pipe has been geometrically simplified as depicted in Fig.1(b). These components, such as the engine, accessories, fan, air-inlet pipe and exhaust pipe, are built in the model. Whereas systems too complex or considered as having a low impact on thermal and flow results have been removed, and supposing no air leaked from the nacelle.

GOVERNING EQUATION

The calculation method of the ventilation cooling performance based on rotor downwash flow employed the steady state, incompressible Navier-Stokes equation as the governing equation, and corresponding turbulent model was attached to solve the turbulent flow. Simultaneously the continuity equation and the energy equation were solved together. In the rectangular coordinate system, the governing equations were expressed as follows:

$$\text{div}(V\phi) = \text{div}(\Gamma \text{grad} \phi) + S \quad (1)$$

Where ϕ is the general variable, which can be the variable of u , v , T , k as needed. V is velocity vector. Γ is the generalized diffusion coefficient corresponding to ϕ , and S is the generalized source term.

BOUNDARY CONDITION

In the simulation, the exhaust gas of the engine was assumed to be ideal gas. Its mass flow rate is m_1 kg/s, and the temperature is t_1 °C. The atmospheric pressure is 9800Pa and the ambient temperature is 308K under hovering and forward flight conditions. The boundary conditions of the engine were determined according to the test data of Engine Ground Stand Test. The nacelle cowl is honey comb structure composite, and it has a low value of heat conductivity. So it is considered as the adiabatic condition. The forward flight velocity was 72m/s and was applied to the front surface of the external flow domain as velocity boundary condition. The downwash flow, namely, axial induced velocity, could be obtained by

the following relational expression which was fitted by the data provided, and applying to the upper surface of the external flow domain.

$$v = -0.003r^5 + 0.1r^4 - 1.34r^3 + 6.22r^2 - 15.99r - 0.80 \quad (2)$$

Where v is the downwash velocity and r is the radial component along the rotor wing.

No slip boundary conditions were applied on all walls of the nacelle, exhaust ejector, exhaust pipe and all accessories. Hence, the flow behavior of near-wall fluid was defined by the standard wall function law, which was a practical option for the near-wall treatments in industrial flow simulations [12].

GENERATION OF GRIDS AND SOLVING STRATEGY

The computational flow domain consists of external flow region underside of the rotor wing, engine nacelle and exhaust pipe. So the computational domain was discretized using Blocked Mesh technique. The nacelle and exhaust pipe domain were meshed with tetrahedral grids and the external flow domain was meshed with hexahedron unstructured grids, as shown in Figure 2. The pre-process was done with ICEM CFD. In order to increase the simulation accuracy, local grid refinements had been achieved in the specific area near the cowl, exhaust ejector and the inlet. In order to reduce time and computer resources necessary, coarser meshes were adopted for the external flow domain than the ones meshed in the nacelle. The error resulted by different grid number was less than 2% when the total number exceeded about 2.2 million. So it was considered that the simulations with different grid number showed a satisfactory mesh-independence.

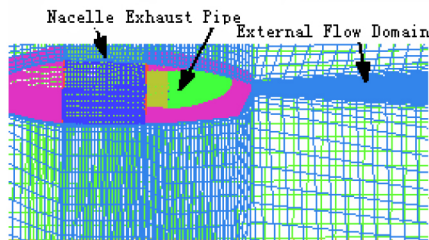


Fig.2. Partial domain mesh model

The finite volume method was employed for the discreteness of the governing differential equations. Thermal Energy Model was applied to calculate heat transfer. Discrete Transfer Model was adopted for the radiative transfer, and the scatter along every direction left out of account. The computation was accomplished on a minitype server. The

simulations required 150 to 200 iterations to obtain reasonable results.

VENTILATION COOLING SCHEME DESIGN

In order to analyze the influence of airflow inlet number, inlet size and inlet position on the ventilation cooling performance of the nacelle, Five single inlet schemes and four double inlets schemes were designed. For all schemes, the nacelle and exhaust ejector have the same configuration and dimension. The illustrations of inlet size and position are listed in table 1. The airflow inlet is drilled in the cowl, and the inlet shape is quadrilateral (Refer to Fig.1). In table 1, Front Upside denotes that the inlet position is located at the front-upper region of the cowl. Front Underside denotes the inlet position is located at the front underside of the cowl. The others descriptions about the position can refer to the relative word meaning. For all double inlets schemes, the inlet1 location is the same as that of A1 scheme.

RESULTS AND DISCUSSION

HOVERING PHASE

Flow field analyses

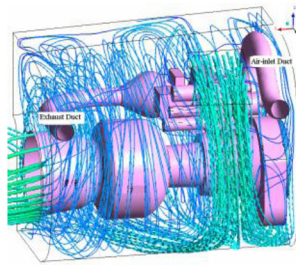
For single inlet scheme, Figure 3 shows the airflow streamlines of A1 and A2 scheme in the nacelle at hover. Streamlines of A3, A4 and A5 scheme are similar to those of A1 scheme. The streamlines reveal the highly complex nature of the flow in the nacelle, and the airflow presented spiral flow encircling the engine shell. As the incoming flow hit the nacelle wall, vortical structures were formed. The velocity was larger at the inlet and outlet of the nacelle, which was not less than 15m/s. However, the velocity in the back space was smaller relatively, and the value was less than 3.5m/s.

Velocity vector at the inlet are presented in Fig.4 for A1, A2 and A5 scheme. Owing to the interference of the fuselage, the downwash flow had an angle relative to the vertical direction at the inlet. So, the inlet position has an important influence on the air coming into the nacelle. In Figure 4(b), the incoming flow entered into the nacelle approximately paralleling the inlet plane, it did some harm to the ventilation cooling of the nacelle. In Fig.4(c), a portion of airflow did not enter into the nacelle at the inlet for A5 scheme. This indicates that increasing inlet size at some location can make against the air coming into nacelle.

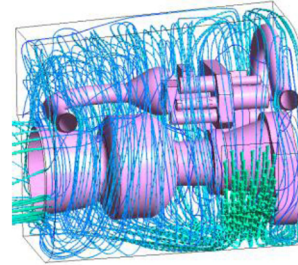
The velocity distributions on YZ plane at $x=7.32\text{m}$ are shown in Fig.5 for A1, A2 and A3 scheme. The plane locates at the turbine zone. As shown in Fig.5, the velocity was higher near the underside, inner side and top wall, but it was lower at the upside of the engine, which was less than 4m/s. As a result, it was difficult that the heat dissipated by the engine was dispersed, and that the temperature of the nacelle should be

Table 1. Illustration of inlet size and position for all schemes

Scheme	Inlet1 Size	Inlet2 Size	Inlet1 Position	Inlet2 Position
A1	300mm×250mm	-	Front Upside	-
A2	300mm×250mm	-	Front Underside	-
A3	300mm×250mm	-	Middle Upside	-
A4	250mm×200mm	-	Same to A1	-
A5	400mm×300mm	-	Same to A1	-
B1	300mm×250mm	150mm×100mm	Front Upside	Back Upside
B2	300mm×250mm	150mm×100mm	Same to B1	Middle Upside
B3	300mm×250mm	150mm×100mm	Same to B1	Back Underside
B4	300mm×200mm	150mm×100mm	Same to B1	Back Topside

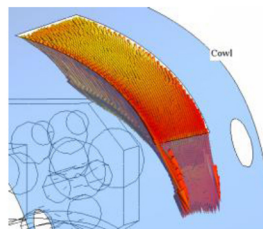


(a) A1 Scheme

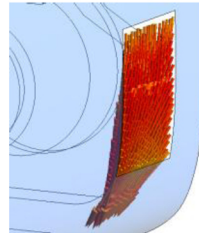


(b) A2 Scheme

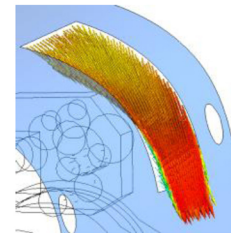
Fig.3. Streamline diagram of airflow in nacelle for A1&A2 scheme in hovering



(a) A1 Scheme



(b) A2 Scheme



(c) A5 Scheme

Fig.4. Velocity vector at inlet for A1, A2 and A5 schemes

higher. The airflow velocity close to the cowl was not less than 6.0m/s for each scheme. High velocity resulted in high heat convection coefficient. So it conducted the cooling the cowl.

For double inlets scheme, Fig.6 presents the airflow streamlines of B1 and B3 scheme in the nacelle at hover. Streamlines of B2 and B4 scheme are similar to those of B1 scheme. As shown in Fig.3 and Fig.6, it was known that the air flow through the inlet1 into the nacelle was similar to the flow corresponding to single inlet scheme and the flow was highly complex. The air flow through the inlet2 showed obvious distinction with its position variation. Because of the inlet2 located at the underside of the cowl, a part of air flowed out of the nacelle consequentially. As fire suppression

requires that no air flows out of the nacelle, B3 scheme can not meet the requirement. Here, the airflow through the inlet2 did not reach the front space of the nacelle for each scheme. As a consequence, it had no contribution for the cooling of the front components when the inlet2 was located at the back side.

Moreover, the velocity distributions were not uniform at the cross section of the exhaust ejector for all schemes. The radius of the curvature of the exhaust pipe was smaller, the velocity at this side was larger. Whereas the radius of curvature was larger, the velocity was smaller.