Fast Dynamic Simulation Analysis of the Thermal Environment of Aircraft Lubricating Oil and Hydraulic Cooling System

Liangliang Liu[1], Yunhan Fu[1], Meng Zhang[1], Xiaokai Zhang[2], and Guanghan Yan[2](✉)

[1] Shenyang Aircraft Design Institute, Electromechanical System Department, Shenyang 110035, China
[2] Dalian University of Technology, Dalian 116023, China
yanguanghan@mail.dlut.edu.cn

Abstract. The continuous increase in the power of airborne electronic equipment and the decrease of the airborne available heat sinks are the inevitable trends of the next generation aircraft. The lack of cabin heat dissipation capacity makes the operating environment temperature of airborne electronic equipment rise and the reliability decrease, thus affecting the combat performance of aircraft. Therefore, it is urgent to study the cooling capacity of the cabin area. In order to obtain the real-time variation of the cabin thermal environment during the whole flight process, the one-dimensional thermal path system of the oil and hydraulic cooling system AMESim is innovatively established to obtain the thermal boundary conditions of the equipment changing with time. Moreover, Fluent was substituted into the three-dimensional simulation analysis of the cabin environment and cabin equipment temperature field, and the simulation results of the transient temperature field and steady-state temperature field were compared. This method can be extended to the external environment under different flight conditions and the internal thermal equipment conditions to quickly and real-time evaluation of different cabin thermal environments. It can provide effective support and input conditions for the environmental thermal design of next generation aircraft cabin areas.

Keywords: Cabin thermal environment · Lubricating oil cooling · Hydraulic oil cooling · co-simulation

1 Introduction

1.1 A Subsection Sample

The next generation aircraft has hypersonic speed, high stealth, high intelligence, and other technical characteristics. The improvement of aircraft performance requires the support of more high-power onboard electronic equipment and more advanced mission systems. Meanwhile, the aerodynamic thermal load cannot be ignored during hypersonic flight, resulting in a large increase in airborne thermal load. However, the availability of heat sinks for the next generation aircraft is declining. On the one hand, the next generation aircraft skins will use composite materials on a larger scale, which do not effectively

Chinese Society of Aeronautics and Astronautics (Ed.): CASTC 2023, LNME, pp. 485–495, 2024.
https://doi.org/10.1007/978-981-99-8864-8_46

radiate heat to the surrounding environment. On the other hand, high stealth performance requires the reduction or elimination of aircraft surface stamping. Adopting new adaptive engine technology will reduce fuel consumption, resulting in the next generation aircraft's traditional ram air heat sink and fuel heat sink being reduced. Therefore, the next generation aircraft onboard equipment faces a serious cooling problem.

However, relevant studies show that over 55% of electronic equipment failures are caused by excessive temperature [1]. The famous 10°C rule points out [2]: The reliability of electronic devices is closely related to the temperature. When the temperature is 70 °C–80 °C, the reliability decreases by 50% for every 10 °C increase. Therefore, the next generation aircraft urgently needs more advanced thermal analysis methods, in order to use limited energy and heat sinks to achieve more precise environmental control.

At present, the CFD numerical simulation method is mainly used to simulate the aircraft cabin's thermal environment. Typical flight status points are selected and the equipment in the cabin is set as the boundary of constant temperature or constant heat flux to calculate the steady-state flow field and temperature field in the cabin. Among them, Gao [3], Wang Lijing [4], Qin [5], Wang [6], and Ma [7] et al. conducted numerical analysis on the flow field and temperature field in a passenger aircraft cabin, passenger aircraft cockpit, helicopter attachment cabin, engine compartment, and engine nacelle. Yang et al. established a CFD numerical calculation model of three-dimensional steady-state fluid-solid coupling heat transfer [8]. Cavage [9] and the multiphase flow model of CFD were used to simulate and calculate the fuel temperature field distribution of the wing tank. Zhang [10] et al. simulated the unsteady variation process of the fuel temperature field inside the tank. The results show that the CFD numerical simulation method has a large number of grids and a long calculation time, and the steady-state calculation results of a single flight state point are much different from the actual.

On the other hand, scholars at home and abroad use a one-dimensional thermal analysis method based on lumped parameters to carry out performance simulation analysis of process systems such as aircraft air circulation refrigeration or fuel thermal management. Among them, Griethuysen et al. discussed in detail the one-dimensional fluid thermal analysis method of aircraft integrated thermal management system [11]. Chang [12] and Gao [13] et al. took the F-22 thermal management system as the prototype to establish the steady-state and dynamic mathematical models of typical components respectively. Zang [14] et al. simulated the performance of an aircraft environmental control system based on AMESim. Jensen [15], German [16], and Li Bo [17] et al. conducted a simulation analysis and test verification of the fuel thermal management system. The results show that the one-dimensional thermal analysis method based on lumped parameters can achieve rapid and dynamic thermal analysis, but it does not have the ability to solve the flow field and temperature field in the cabin.

Under the influence of high ultrasonic aerothermal, the inner wall temperature of the next generation aircraft will reach 100 °C, and the heat will be transferred to the equipment in the cabin utilizing heat conduction, radiation, and convection. At the same time, both the aircraft structure and the equipment in the cabin have large heat sinks, and the heating power of most equipment in the cabin is constantly changing with the flight envelope. Therefore, it is necessary to comprehensively consider the cabin structure, cabin air and cabin equipment, and carry out integrated dynamic thermal analysis along the envelope. To further accurately evaluate the cabin thermal environment, optimize the environmental control scheme, and ensure the safe and reliable operation of the cabin equipment.

In this paper, a lumped parameter one-dimensional dynamic thermal analysis method is proposed to obtain the temperature of equipment or thermal system in cabin. These boundary conditions are transferred to CFD numerical simulation of the cabin thermal environment in real-time. At the same time, the surface heat flux of the equipment is obtained by CFD numerical simulation, which is transferred to the one-dimensional thermal analysis of the equipment in the cabin in real-time as the boundary condition. Then, through the co-simulation method of the two, the integrated rapid dynamic thermal analysis function of the aircraft thermal environment and the equipment in the cabin is realized. The results of an aircraft engine accessory compartment where the oil-oil radiator and the oil-hydraulic radiator are located show that the method is fast and the calculation results are more real.

2 Basic Principle and Calculation Method

2.1 Mechanism of Heat Transfer in Cabin Thermal Environment

The cabin thermal environment is the result of the multilevel coupling of the multilevel physical fields. From the perspective of heat generation and transmission mechanism, the formation of an aircraft thermal environment includes five thermal physical processes, as shown in Fig. 1. Aerodynamic heating or cooling of skin by air flow outside the aircraft. Solar radiation heating of aircraft skin and radiation heat transfer between skin and ground and space. Unsteady heat conduction of skin under the coupling effect of thermal environment inside and outside aircraft. The unsteady radiation-convection-thermal coupling heat transfer between the inner surface of skin, the components and equipment in aircraft cabin and the air flow in aircraft cabin.

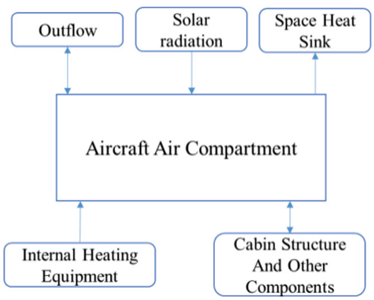

Fig. 1. Thermal Environment of Aircraft Cabin.

2.2 Fluid Governing Equation

In order to obtain the characteristic parameters of the flow field and temperature field inside the cabin, it is necessary to solve the continuity equation, momentum equation, and energy equation simultaneously. Due to the existence of sub-current flow inside the cabin, it is necessary to increase the solution of the turbulent transport equation to form a closed loop.

Continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

where u, v, w is the velocity vector along the x, y, z directions.

Momentum equation:

$$\rho\frac{\partial \mathrm{u_i}}{\partial \mathrm{t}}+\mathrm{u}\frac{\partial \mathrm{u_i}}{\partial \mathrm{x}}+\mathrm{v}\frac{\partial \mathrm{u_i}}{\partial \mathrm{y}}+\mathrm{w}\frac{\partial \mathrm{u_i}}{\partial \mathrm{z}}=\rho F_{\mathrm{x}}-\frac{\partial \rho}{\partial \mathrm{x}}+\mu\left(\frac{\partial^2 \mathrm{u_i}}{\partial \mathrm{x}^2}+\frac{\partial^2 \mathrm{u_i}}{\partial \mathrm{y}^2}+\frac{\partial^2 \mathrm{u_i}}{\partial \mathrm{z}^2}\right) \tag{2}$$

where, $u_i, i=1,2,3$ are velocity vectors along the x, y and z directions respectively; ρ is the density; F_x is the component of the force in the x direction; μ is the dynamic viscosity.

Energy equation:

$$\frac{\partial^2 \mathrm{t}}{\partial \mathrm{x}^2}+\frac{\partial^2 \mathrm{t}}{\partial \mathrm{y}^2}+\frac{\partial^2 \mathrm{t}}{\partial \mathrm{z}^2}=0 \tag{3}$$

where t is the temperature of the fluid micro clusters.

Turbulent flow equation:

$$\rho\frac{\mathrm{dk}}{\mathrm{dt}}=\frac{\partial}{\partial \mathrm{x_i}}\left[\left(\mu+\frac{\mu_\mathrm{t}}{\sigma_\mathrm{k}}\right)\frac{\partial \mathrm{k}}{\partial \mathrm{x_i}}\right]+\mathrm{G_k}+\mathrm{G_b}-\rho\varepsilon-\mathrm{Y_M} \tag{4}$$

$$\rho\frac{\mathrm{d}\varepsilon}{\mathrm{dt}}=\frac{\partial}{\partial \mathrm{x_i}}\left[\left(\mu+\frac{\mu_\mathrm{t}}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial \mathrm{x_i}}\right]+\rho c_1\mathrm{S}\varepsilon-\rho c_2\frac{\varepsilon^2}{\mathrm{k}+\sqrt{\mathrm{v}\varepsilon}}+\mathrm{c_{1\varepsilon}}\frac{\varepsilon}{\mathrm{k}}\mathrm{c_{3\varepsilon}}\mathrm{G_b} \tag{5}$$

where $c_1=\max\left[0.43,\frac{\eta}{\eta+5}\right]$; $\eta=SK/\varepsilon$, G_k represents the turbulent kinetic energy generated due to the average velocity gradient; G_b represents the generation of; Y_M represents the effect of compressible turbulent pulsation expansion on the total dissipation rate; σ_k and σ_ε are the turbulent Prandtl number of turbulent kinetic energy and its dissipation rate respectively.

2.3 Co-simulation Method

A one-dimensional and three-dimensional co-simulation method based on AMESim and Fluent software is proposed. The real-time temperature of the heating equipment at the state point of the whole flight profile is obtained by AMESim modeling. These are then input into the Fluent model as boundary conditions. Thus, the cabin temperature environment of thermal load at the point of full profile state is obtained (Fig. 2).

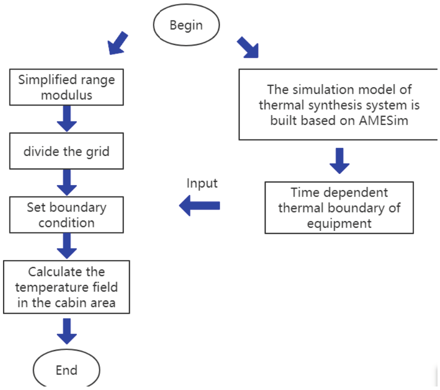

Fig. 2. The Computing process.

3 One-Dimensional Thermal Analysis of Cabin Thermal System

3.1 Thermal System Composition

The cabin heating equipment discussed in this paper is oil-oil radiators and oil-hydraulic radiators in airborne thermal integrated systems. The oil-oil radiator is distributed in the oil cooling subsystem of the generator. The hot oil from the left and right generators is filtered and the fuel oil is used to reduce the oil temperature. The oil-hydraulic oil radiator is installed in the oil return circuit of the hydraulic piston pump. The oil return circuit of the shell is powered by the hydraulic pump. The oil-hydraulic oil radiator is used for heat exchange with the fuel system. The fuel used for heat dissipation in the fuel system comes from the oil tank 3. After being pressurized by the oil supply pump, the fuel first flows through two oil-hydraulic oil radiators, and then through two oil-lubricating oil radiators. The hot oil after absorbing heat enters the engine for consumption, and the fuel consumed by the engine returns to the oil tank 1 through the hot return oil path. The system schematic diagram is shown in Fig. 3.

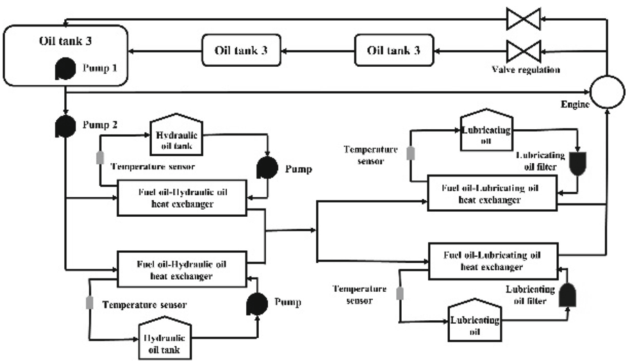

Fig. 3. Schematic Diagram of Thermal Management System.

3.2 Simulation Model

AMESim software is an advanced modeling and simulation platform of Siemens engineering system. The software can realize the joint modeling and simulation of mechanical, hydraulic, pneumatic, thermal fluid, electromagnetic, control, and other multi-disciplinary fields. It is also suitable for modeling of multi-system coupling airborne electromechanical system [8, 18].

The one-dimensional simulation model of thermal synthesis system is built based on AMESim. The generator oil cooling system and hydraulic heat exchange system are packaged in super components. The system simulation model is shown in Fig. 4-5.

The corresponding simulation models of the main components of the thermal synthesis system in AMESim are shown in Table 1. The thermal environment under each section of the system is the actual flight process data.

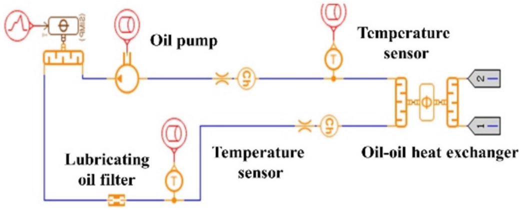

Fig. 4. Simulation Model of Generator Oil Slip Cooling System.

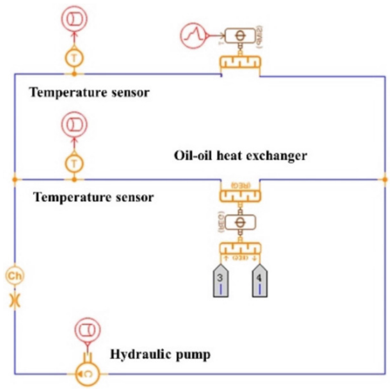

Fig. 5. Simulation Model of Hydraulic Heat Exchange System.

3.3 Simulation Result

According to the flight altitude and Mach number in the assumed flight profile from 1 s to 2000 s as boundary conditions, a thermal synthesis simulation model is used in AMESim. Based on this, the temperature data of the hot and cold edge inlet and outlet of the oil-oil radiator and oil-hydraulic oil radiator are calculated with time under assumed flight profile. See Fig. 6-7.

Table 1. Simulation Model Table of Main Components Corresponding.

Component	Simulation model
Temperature sensor	A hot fluid temperature sensor module
Pump	Hot fluid temperature, mass flow module
Governor valve	Adjustable pneumatic orifice module
Thermal equipment	Hot power module + Hot fluid half heat exchanger module
Fuel oil	Hot fluid properties module
Hydraulic oil	Hot fluid properties module
Grease	Hot fluid properties module
Air	Gas attribute module
High-temperature air source	Pneumatic pressure and temperature source module
Oil-Hydraulic oil radiator	Hot fluid heat exchanger module + Hot fluid calculation module
Oil-Oil radiator	Hot fluid heat exchanger module + hot fluid calculation module
Oil-Air radiator	Hot fluid heat exchanger module + hot fluid calculation module
Oil–Air radiator	Pneumatic/thermal fluid heat exchanger module + thermal fluid calculation module
Mixed Vegetables	Hot fluid tee joint

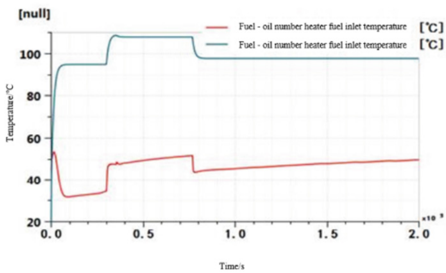

Fig. 6. Simulation Results of Fuel-oil Radiator.

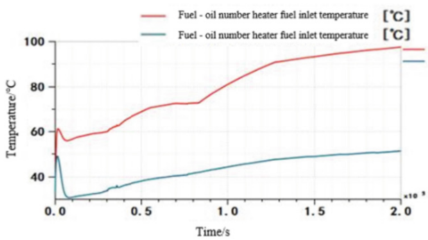

Fig. 7. Simulation Results of Fuel- Hydraulic fluid Radiator.

4 CFD Numerical Simulation of Cabin Thermal Environment

4.1 Mesh Model

Simulate a typical cabin area of an aircraft. The upper and lower surfaces are aircraft skin (affected by aerodynamic heating), and four heating devices are distributed in the cabin. These include two oil-oil radiators for the generator oil cooling system and two oil-hydraulic oil radiators for the hydraulic heat exchange system, all of which are of actual aircraft heat exchanger sizes. ICEM software was used to divide and optimize the computing domain of the calculated cabin area, oil-hydraulic oil radiator and oil-oil radiator models. The grid is encrypted near the radiator wall, the number of grids is 6.67 million, and the grid quality is no less than 0.2, as shown in Fig. 8.

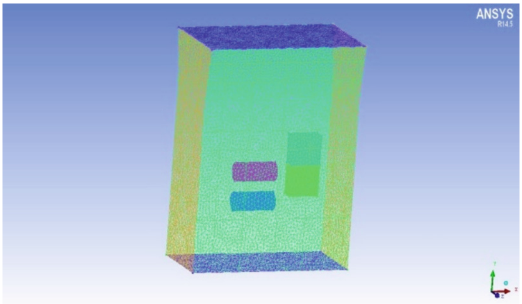

Fig. 8. Computational Grid.

4.2 Boundary Conditions

The inlet and outlet boundary conditions are set according to the actual flight state and working condition. The outlet pressure is set to the outside atmosphere, and the left and right bulkheads are set to 363 K. For the rest of the cabin, the surface temperature is set to 293.15 K, the cooling side flow of the radiator is set to 3000 L/h, and the hot side flow is set to 1500 L/h. In the transient simulation process, the temperature curve in Fig. 6 and Fig. 7 of the one-dimensional calculation results of AMESim is fitted into the temperature relation with time. Then write UDF, load as the heat sink inlet temperature conditions. The flight altitude and flight Mach number in the flight state boundary of aircraft are set according to the assumed flight profile.

4.3 Simulation Results

When calculated using the joint simulation method, the temperature field in the simulated cabin area varies with time under the assumed flight profile as shown in Figs. 9, 10 and 11. Due to the heating by the radiator and bulkhead, the air inside the compartment area experiences a continuous temperature increase with time. When the temperature of the early bulkhead is higher, the influence of the adjacent compartment on the heating of the middle compartment is greater. In the later period, with the temperature of the heating equipment gradually rising, its influence on the cabin environment also began to increase.

The steady-state temperature field in the cabin area is calculated according to the traditional cabin area steady-state temperature field simulation method, and the cabin heat-generating equipment is set to a constant temperature of 70 °C. The steady-state temperature field in the cabin area is shown in Fig. 12.

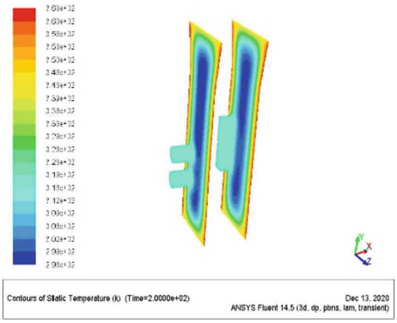

Fig. 9. Cloud diagram of temperature distribution in Aircraft cabin (t = 600 s).

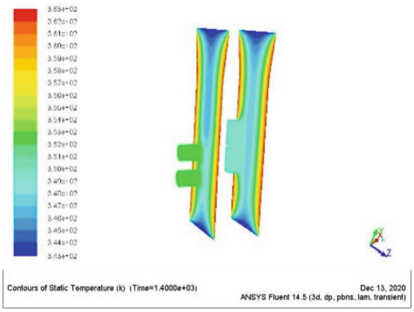

Fig. 10. Cloud diagram of temperature distribution in Aircraft cabin (t = 1400 s).

Some conclusions can be drawn by comparing the simulation results of transient temperature field with those of steady-state temperature field. The steady-state calculation can obtain the temperature distribution of the cabin under specific working conditions. However, in the real world, because the temperature of bulkhead and heating equipment is constantly changing, so is the temperature of the cabin. By transient calculation, the cabin temperature distribution under different flight profiles can be obtained when the heat source changes with time. At this point, it is obvious that the transient calculation is more realistic and objective for the temperature environment assessment of the cabin area under the full profile, and can realize the real-time dynamic demand capture of the aircraft cabin thermal design.

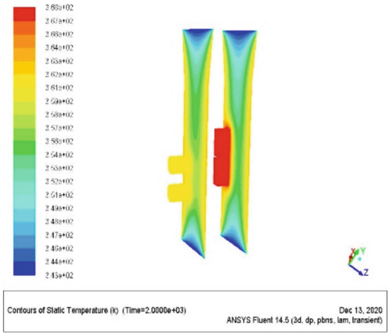

Fig. 11. Cloud diagram of temperature distribution in Aircraft cabin (t = 2000 s).

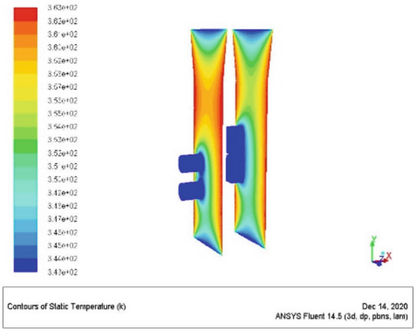

Fig. 12. Cloud diagram of temperature distribution in Aircraft cabin.

5 CFD Numerical Simulation of Cabin Thermal Environment

In this paper, the temperature field variation with time in the lower cabin area of the flight profile is studied. Then, the co-simulation model of a three-dimensional cabin temperature field based on AMESim and Fluent platform is established. The real-time simulation results of the cabin temperature field varying with time under flight profile are obtained and compared with the traditional steady-state calculation of the cabin temperature field. By comparing the results, it is found that the one-dimensional thermal synthesis system and the three-dimensional co-simulation model of cabin temperature field are more realistic and objective to evaluate the temperature environment of the cabin under the full profile. The co-simulation model can capture the real-time dynamic requirements of aircraft cabin thermal design better.

The purpose of this paper is to provide a fast dynamic simulation method for the thermal environment of the aircraft cabin areas, which can be extended to cabin area models of different structural forms, different heat-generating devices, and different flight profiles. It provides effective support and input conditions for the next generation aircraft cabin area thermal design research.

References

1. Janicki, M., Mapieralski, A.: Modeling electronic circuit radiation cooling using analytical thermal model. Microelectron. J. **31**, 781–785 (2001)
2. Ozmat, B.: Interconnect technologies and the thermal performance of MCM. Components Hybrids & Manufacturing Technol. IEEE Trans. **15**(5), 860869 (1992)
3. Gao, F.: Numerical simulation and comfort evaluation of flow field in aircraft cabin. Aeronautical Computing Technique **49**(3), 20–23 (in Chinese)
4. Wang, L., et al.: Airflow thermal simulation and com-fort evaluation of commercial airliner. J. Beijing University of Aeronautics and Astronautics **36**(12), 1436–1439 (2010) (in Chinese)
5. Qin, Y.: Numerical simulation study on flow field and temperature field of accessory nacelle on a helicopter. Helicopter Technique **142**(2), 19–23 (in Chinese)
6. Wang, Y.: Numerical simulation of nacelle flow and temperature field using CFD. Aircraft Design **33**(6), 1620 (2013) (in Chinese)
7. Ma, M.: Computation of Flow and Heat Transfer in Aeroengine Nacelle. Northwestern Polytechnical University, Xian (2007) (in Chinese)
8. Yang, Q., et al.: Study on temperature field calcula-tion of dry air flying skin of hot air antiicing system based on fluid-solid coupling heat transfer. J. Aerodynamics **34**(6), 721–724 (2016) (in Chinese)
9. William, M.: Cavage modeling in-flight inert gas distribution in a 747 center wing fuel tank. In: 35th AIAA Fluid Dynamics Conference and Exhibit, AIAA 2005–4906
10. Zhang, X., et al.: Analysis of fuel temperature change in advanced fighter aircraft during supersonic cruise .J. Aerodynam **25**(2), 258–263 (2010) (in Chinese)
11. Griethuysen, J.: A modeling approach for integrated thermal management system analysis of aircraft. SAE Aerospace Power Systems Conference Proceedings (1997)
12. Chang, S., Yuan, M., Yuan, X.: Steady state simulation of airborne integrated thermal management system. J. Beijing University of Aeronautics and Astronautics **34**(7), 821–824 (2008) (in Chinese)
13. Gao, F., Yuan, X.: High performance combat aircraft fuel heat management system. J. Beijing University of Aeronautics and Astronautics **35**(11), 13531356 (2009) (in Chinese)
14. Zang, X., et al.: Performance simulation and improvement of new type aircraft environment control system base on AMESim software. Aircraft De-sign **38**(4), 5660 (2018) (in Chinese)
15. German, B.J.: Tank heating model for aircraft fuel thermal systems with recirculation. J. Propuls. Power **28**, 204–210 (2012)
16. Jensen, D.L.: Analysis of a Boeing 747 aircraft fuel tank vent system. J. Aircr. **38**(5), 828–834 (2001)
17. Li, B., et al.: Simulation and experimental verification of aero-engine fuel thermal management system **32**(5), 29–34 (2019) (in Chinese)
18. Fu, Y., Qi, X., et al.: System Modeling and Simulation Reference Manual. Beijing University of Aeronautics and Astronautics Publisher, Beijing (2011). (in Chinese)