Research Article

Numerical Study of Air Distribution System for a Passenger Aircraft Based on 1D and 3D Coupling

Yi Tu,1 Xuanren Chen,2 Liang Wang,1 Liang Yin,1 Qiuyun Zheng,1 and Yu Zeng3

1Hunan University of Arts and Sciences, Hunan Key Laboratory of Distributed Electric Propulsion Vehicle Control Technology, Changde 415000, China
2AVIC Integrated Equipment Co., Ltd, Beijing 102206, China
3Beihang University, Beijing 100191, China

Correspondence should be addressed to Yu Zeng; zengyu@buaa.edu.cn

Received 22 April 2022; Accepted 5 September 2022; Published 20 September 2022

Academic Editor: Juan Du

Copyright © 2022 Yi Tu et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

The air distribution system is one of the key systems for ensuring the safety and comfort of the passenger aircraft. The large number of components poses a severe challenge to the efficiency, accuracy, and convergence of system simulation. For the 3D CFD method, the complex and large number of meshes leads to low computational efficiency. The 1D simulation has high efficiency, but the local accuracy is inadequate. To solve this problem, the 1D Fluid System Simulation Software Flomaster with the linearization algorithm is used to establish the system-level model. The flow characteristic curve is obtained by the CFD (computational fluid dynamics) method for the local irregular ducts and introduced into the 1D system model for iteration. The effects of local flow resistance and thermal insulation on the flow distribution and thermal performance of the system are studied using this simulation model. The results show that the local flow resistance affects the total flow and uniformity of the cabin distribution, the improvement of insulation performance is weakened with increasing insulation layer thickness, and the application of thicker insulation layers on the downstream pipeline is more advantageous to limit the maximum temperature change of the conditioned air.

1. Introduction

The air distribution system supplies fresh air with adjusted parameters to the cabin and takes away the thermal load generated by electronic equipment and personnel, to ensure the temperature, humidity, and wind velocity in a comfortable range for the high passenger density cabin environment of the aircraft [1, 2]. The air distribution system is generally composed of multiple subsystems according to the main branch at the outlet of the mix manifold, which supply the air with the required mass flow rate to each cabin area. The design of the air distribution system should first meet the fresh air requirement of each compartment in the cabin to ensure the safety of passengers and the uniformity of the air flow rate of each air outlet to ensure thermal comfort and fresh air demands of different cabin regions [3]. There are many factors that affect the uniformity of air distribution, including the diameters of main and branch pipes, the number of branch pipes, the size of orifices, and the flow resistance caused by local deformation of the pipeline. To ensure the temperature control ability of the aircraft cabin, it is also necessary to optimize the thickness of the insulation layer on the system pipes to guarantee that the conditioned air still has sufficient heating or cooling capacity when it reaches the branch outlets. However, increasing the insulation thickness of the pipeline will increase the total weight of the system, which is a sensitive factor in aircraft design. The engineering design of the system should reduce the total weight of the system as much as possible while achieving basic insulation performance.

Most of the research work on air distribution systems focuses on the effects of air supply and exhaust modes on thermal comfort and energy savings in the aircraft cabin. Liping Pang et al. proposed an improved air distribution...
system with personalized outlets placed at the bottom of the baggage hold. Experiments and simulations revealed that this improved pattern had the potential to save energy while maintaining the same level of health and air flow quality [4]. A.M. Farag et al. compare and analyze the simulated cabin environment with mixing, underfloor displacement, and personalized ventilation systems to improve air quality in the aircraft cabin of the economy section of a Boeing 767 airplane using computational fluid dynamics techniques [5]. Zhang conducted experimental research to obtain the local mean age of air, temperature, and velocity distribution under mixing ventilation and displacement ventilation methods, using trace gas, thermocouples, and an ultrasonic anemometer measurement system. The result shows that the displacement ventilation shows high ventilation efficiency and high energy efficiency [6]. You evaluated the effectiveness of three different ventilation systems in controlling the transport of contaminants and maintaining thermal comfort in the cabin using the CFD method. The results showed that the personalized ventilation system performed the best in maintaining thermal comfort in the cabin and can also reduce the risk of infection by air [7]. Cao et al. studied the influence of different cabin geometric simplification schemes and the selection of the turbulence model on the accuracy of the results of the CFD analysis from the perspective of the simulation modeling method [8].

The above studies aim at the impact of cabin ventilation on personnel comfort. As the source of the cabin air supply, the performance of the air distribution pipe system also has a great impact on cabin comfort and energy economy. There are relatively few studies on this aspect. In the early stage, some research on aircraft air systems was carried out based on MATLAB/Simulink [9–11].

Subsequently, researches of aircraft air systems based on Aspen Plus [12], AMESim [13, 14], and other software gradually appeared, but the objectives of these researches are only limited to the air cycle machine with only a small number of components. In these methods, the mathematical model of each component and the variable transfer relationship between components are constructed, and the traditional numerical solution is used to solve these equations. CFD analysis is also used to study the aircraft air duct system, but the research object is the system model composed of a small number of pipes with simple geometric structures [15], or a single local pipe [16]. However, the air distribution system includes a large number of pipes, valves, joints, orifices, pressure sources, and local flow resistances. It is difficult to reach convergence in this system-level analysis owing to the large number of components, and traditional methods based on Matlab, AMESim, or 3D CFD are not the best choice for this system-level modeling considering computational efficiency and
convergence. The one-dimensional (1D) thermal fluid analysis software Flomaster is widely used in complex fluid systems analysis, such as fuel, oxygen, hydraulic, and air systems [17–19] and is based on the linearization coefficient method of the models and possesses superior convergence properties compared to other numerical solution software.

The disadvantage of 1D system-level simulation is that the modeling process depends on the mathematical expression of the standardized model. However, it is difficult to express the flow characteristics of nonstandard components and irregular structures through mathematical models. If the entire system is simulated using the 3D CFD method, the number of meshes will be large. It will cause low computational efficiency, which is unacceptable in engineering analysis. One of the main works of this paper is to find a numerical method with high efficiency, good convergence, and fine accuracy in the flow and thermal performance of the air distribution system of passenger aircraft. Based on this method, numerical analyses will be performed to study the effect of local resistance on flow distribution and the effect of insulation on the overall thermal performance of the pipeline.

2. System Configuration

Figure 1 illustrates the schematic of the passenger aircraft air distribution system of the passenger aircraft. The air distribution system is used to convey conditioned air to each compartment of the aircraft cabin in a specific proportion. The cooling air from the packs and the recirculated air from the cabin were completely mixed in the mix manifold. The mix manifold outlets connect to the cockpit distribution and left and right cabin distribution main pipes, through which the conditioned air is delivered to various compartments such as the cockpit, the forward cabin, the afterward cabin, the forward galley/lavatory, and the afterward galley/lavatory. The branch pipes are connected to the main pipes through junctions in the location area. Orifices were frequently assembled to branch pipes to balance the flow distribution at the outlets. The entire pipeline system is covered with thermal insulation material to reduce heat loss as conditioned air is transported from the mix manifold to the end of the pipeline.

3. Main Component Model Identification

Fluid flow and heat transfer analysis was performed using the Simcenter Flomaster 2020 program and the ANSYS-Fluent (CFD) program. Cabin and cockpit distribution pipe systems were modeled in Flomaster. The CFD program was used when complex duct geometry and high turbulent flow conditions were involved. Flomaster offers cylindrical ducts, transitions, bends, junctions, and orifices as standard definition components to create simulation models. The mathematical expression of the core components of the system is as follows.

3.1. Pipe Model. The pipeline is the main contributor to the flow resistance of the system. The Darcy formula is used to calculate the pipeline pressure loss as shown in the following equation:
\[
\Delta P = f \frac{L \rho V^2}{d^2}, \quad (1)
\]

where \( L \) is the length of the pipe, \( d \) is the hydraulic diameter, \( V \) is the fluid velocity, \( \rho \) is the fluid density, and \( f \) is the Darcy friction factor calculated by the Colebrook-White model as shown in the following equation:

\[
f = \left\{ \begin{array}{ll}
\frac{64}{Re} & \text{for } Re \leq 2000 \\
0.25 & \text{for } Re \geq 4000 \\
\left[ \frac{\log \left( \frac{k}{3.7d} \right) + \left( 5.74/Re^{0.8} \right)^2}{\left( \frac{Re}{2000} - 1 \right) \times 64 + \frac{0.5 - 0.25 \times Re/2000}{\left[ \log \left( \frac{k}{3.7d} \right) + \left( 5.74/Re^{0.8} \right) \right]^2}} \right] & \text{for } 4000 > Re > 2000
\end{array} \right.
\]

(2)
where \( k \) is the surface roughness of the pipe material and the appropriate value of the smooth pipe in this analysis is 0.0025 mm.

The surface of the air distribution pipe is covered with thermal insulation materials. Four parts of thermal resistance should be considered when analyzing the thermal performance of the pipe, as shown in Figure 2, including the convective heat transfer resistance between the internal wall of the pipe and the working medium, the thermal resistance of the pipe wall, the thermal resistance of the insulation layer, and the convective heat transfer resistance between the external wall of the insulation layer and the environment. The total heat transfer resistance is calculated by the following equation:

\[
R = \frac{1}{\lambda_w \delta_w} + \frac{1}{\lambda_e \delta_e} + \frac{1}{h_e} + \frac{1}{h_i}, \tag{3}
\]

\[
Nu = 0.023 \, Re^{0.8} \, Pr^{0.4}, \tag{4}
\]

where \( \lambda_w \) is the thermal conductivity of the pipe material and carbon epoxy was chosen as the pipe material with \( \lambda_w = 6.93 \, \text{W/m-K} \); \( \lambda_e \) is the thermal conductivity of the insulation material, and melamine was used as the insulation layer with \( \lambda_e = 0.038 \, \text{W/m-k} \); \( \delta_w \) is the wall thickness of the pipe; \( \delta_e \) is the thickness of the insulation layer; \( h_e \) is the external heat convection coefficient of the outer surface of the insulation layer, which is defined by the engineer according to the air convection environment around the pipe, where \( h_e = 6 \, \text{W/m}^2\cdot\text{K} \) was set for the natural convection environment outside the pipeline; and \( h_i \) is the internal heat convection coefficient of the pipe, which is calculated by the Dittus-Boelter correlation as Equation (4).

3.2. Orifice. Orifices are used to balance the air distribution at the outlets of the pipe system; the pressure drop through this component is calculated by the following equations:

\[
\Delta P = K_o C_o C_{Re} \frac{\rho V^2}{2}, \tag{5}
\]

\[
K_o = \left[ 1 - \left( \frac{d_o}{d} \right)^2 C_c \right]^2 \frac{1}{(d_o/d)^2 C_{Re}^2}, \tag{6}
\]

where \( K_o \) is the flow loss coefficient (test data provided by Flomaster program) and \( C_o \) is the correction factor depending on the type of orifice (test data provided by the Flomaster program). As for the sharp-edged orifices used in this analysis, \( C_o = 1 \). \( C_c \) is the contraction coefficient (test data provided by the Flomaster program). \( d_o \) is the diameter of the orifice and \( d \) is the diameter of the pipe. \( C_{Re} \) is the laminar flow correction coefficient defined as an experimental curve in the Flomaster program, and \( C_{Re} = 1 \) for turbulent flow.
3.3 Irregular Ducts. The geometric shape of typical irregular ducts is shown in Figure 3. These ducts are characterized by obvious changes in the shape of the duct cross-section, structural distortion, or bifurcation, resulting in local resistance and affecting the flow distribution of the entire system. Since the geometry of this type of pipe is irregular, there is no standard mathematical model to predict the relationship between pressure drop and flow rate. 1D and 3D co-simulation has been considered a method for such cases. However, due to the long calculation time of the 3D CFD program, the iteration efficiency of the co-simulation program will be reduced, which is unacceptable in the process of engineering design.

For the whole system, these irregular-shaped ducts only occupy a few of hundreds of pipe elements; the surface area is small compared to the total surface of all system components. In addition, they are also covered with thermal insulation. Consequently, the heat loss of fluid through these irregular ducts can be ignored. In this section, we focus on the pressure drop as air flows through these irregular ducts. In the possible range of the operating flow rate of these irregular ducts, the relationships between the pressure drop and the volume flow rate are calculated using the CFD method. Using these CFD analysis data, the characteristic curve of pressure drop versus volume flow rate \( \Delta P - Q \) is formed and referenced in the system for iteration by the special system level analysis component.

3.3.1 Numerical Method and Boundary Conditions. The computational fluid dynamics software ANSYS Fluent solver was used to solve the Navier-Stokes equations, and the realizable \( k-e \) model is selected as the turbulence model. When all residual coefficients are less than \( 10^{-5} \) and the weighted average pressure monitoring value of the inlet plane is stable,
4. System Model Development

The system-level simulation model is shown in Figure 6. The entire system is composed of 388 components, most of which can be modeled using standard components in the Flomaster program, such as pipes, bends, control valves, restrictors, and transition joints. These standard components of the Flomaster program are supported by highly reliable engineering data and mathematical models to ensure the accuracy of the simulation analysis. The flow source component with constant mass flow rate and temperature is used to simulate the air supply from the cooling pack. The standard rigid pipe component was used to simulate the cylindrical pipe. The pressure drop through the restrictor, ΔP, is calculated using the orifice component in the Flomaster program. The standard valve component is used to simulate the regulating valve. The standard fan component is used to simulate the recirculation fan. The transition joint used for connecting main pipes of different diameters is simulated by the transition component in the Flomaster program. Standard rigid pipes of large diameter are used to simulate the large volume cabin, and their common feature is that there is almost no pressure drop when the fluid flows through them. The cabin pressure is determined by the opening position of the outflow valve, the exhaust flow rate, and the external atmospheric pressure. The irregular ducts 1, 2, and 3 are located on the main pipe of the afterward cabin and the cockpit, respectively. The existence of irregular ducts is due to the limitation of the structural layout space that the aircraft structural can provide for the main pipe of the air distribution system. For this kind of nonstandard duct, the discrete loss component of the Flomaster program can be used to put the ΔP – Q curve obtained by CFD simulation into the system for iteration. However, because of the large number of components and branches, the convergence of system-level simulation still poses a challenge. The entire system modelling process should be conducted level by level according to the sequence of branches, subsystems, and system.

5. Results and Discussion

The uniformity of the cabin flow distribution and the thermal insulation performance of the pipeline are the key issues of the air distribution system design, which are used to ensure the comfort of passengers and the rapid heating and cooling capacity of the aircraft temperature control system. In this study, the absolute value of the mass flux
deviation among all air outlets should not exceed 5%, the temperature rise caused by the cold loss along the conditioned air pipeline shall not exceed 3°C in the temperature pull-down case of the aircraft cabin in hot days, and the temperature drop caused by the heat loss along the conditioned air pipeline of the conditioned air along the pipeline should not exceed –5°C in the temperature pull-up case of the aircraft cabin in cold days.

There are many factors that affect the air distribution and thermal insulation performance of the entire air distribution system. This paper focuses on the effect of local resistance caused by irregular ducts in the main pipeline on air distribution uniformity and the effect of the thickness and layout of the thermal insulation layer of the air pipe network on the thermal insulation performance, to provide theoretical and methodological references for the subsequent engineering design of the aircraft air distribution system.

5.1. Effect of Local Flow Resistance on Flow Distribution. The volume of locally deformed pipes in this air distribution pipe system is typically very small compared to the total volume of the entire system pipes. However, whether this part of the flow resistance can be ignored in the design of flow distribution uniformity is a key issue that has attracted the attention of engineers. This analysis will focus on the effect of local resistance on the flow distribution of the entire system. In this analysis, the relative deviation of mass flow rate defined by \((\dot{m}_{\text{max}} - \dot{m}_{\text{min}})/\dot{m}_{\text{min}}\) is used to evaluate the uniformity of air distribution at each air supply point.

5.1.1. Air Distribution of the Cockpit. The irregular duct 3 belongs to the pipeline of the cockpit subsystem as shown in Figure 6. Three discrete loss components were used to simulate the behavior of fluid that flows through these irregular ducts. The \(\Delta P - Q\) curves obtained by the CFD method (Section 3.3.3) were used as the input data to describe the variation of pressure drop versus volume flow rate.

As shown in Figure 7, the air distribution system was first designed to balance the flow rate at all the air outlets without considering the local pressure loss of the irregular duct, and the maximum relative deviation of the mass flow rate at the air outlet is reasonably controlled to 3.14%, which meets the design requirements that the relative deviation does not exceed 5%. However, as shown in Figure 8, the maximum relative deviation reaches 9.49% when these parts of the local flow resistance are considered in the analysis. This result does not meet the design requirements for the uniformity of the flow distribution. In addition, it also affects the mass flow rate at each local outlet point of the cockpit. The average mass flow rate of air outlets in the cockpit is reduced from 0.0264 kg/s to 0.0251 kg/s.

5.1.2. Air Distribution of the Cabin. The irregular ducts 1 and 2 are located on the main pipes of the afterward cabin, both of which have a single inlet and a single outlet. Each of them can be simulated using one discrete loss component with its \(\Delta P - Q\) curve as the input data to describe the variation of pressure drop versus volume flow rate. Figures 9 and 10 show the mass flow rates and relative deviations of the air supply outlets in the cabin under the conditions of ignoring and considering the local flow resistance.

Similarly, the air distribution system was firstly designed to balance the flow rate at all outlets of the cabin without considering the local pressure loss of the irregular duct, and the maximum relative deviation of the mass flow rate at the air outlet is reasonably controlled to 3.09%, as shown in Figure 9, which meets the requirements of the design that the relative deviation does not exceed 5%. However, as shown in Figure 10, the maximum relative deviation reaches 7.24% when these parts of the local flow resistance are considered in the analysis. The average mass flow rate of air outlets in the afterward cabin is reduced from 0.0152 kg/s to 0.0149 kg/s.

From this comparative study, it can be seen that the local flow resistance of the air distribution pipeline system will have an impact on the uniformity of the distribution of the system and the total air mass flux of the local compartment. This is because the mass flow rate of the main pipe is high and the flow resistance caused by local deformation is a non-negligible value compared to the global resistance, which will increase the flow rate of the upstream distribution pipe and reduce the flow rate of the downstream distribution pipe.

5.2. Effect of the Insulation Configuration on Duct System Thermal Performance. The air distribution pipeline system must reduce the heat loss of the air along the pipeline by thermal insulation measures to ensure that the conditioned

| No.  | \(\delta_{\text{up}}/\text{mm}\) | \(\delta_{\text{down}}/\text{mm}\) | \(m/\text{kg}\) | \(T_{\text{in}}/°\text{C}\) | \(T_{\text{amb}}/°\text{C}\) | \(P_{\text{out}}/\text{pa}\) | \(T_{\text{in}}/°\text{C}\) | \(T_{\text{amb}}/°\text{C}\) | \(P_{\text{out}}/\text{pa}\) |
|------|-------------------------------|-------------------------------|-----------------|------------------|------------------|-----------------|------------------|------------------|------------------|
| Case1| 18                            | 18                            | 12.11           | 43               | 0                | 101325          | 14               | 30               | 101325          |
| Case2| 15                            | 15                            | 9.85            | 43               | 0                | 101325          | 14               | 30               | 101325          |
| Case3| 12                            | 12                            | 7.69            | 43               | 0                | 101325          | 14               | 30               | 101325          |
| Case4| 9                             | 9                             | 5.63            | 43               | 0                | 101325          | 14               | 30               | 101325          |
| Case5| 6                             | 6                             | 3.66            | 43               | 0                | 101325          | 14               | 30               | 101325          |
| Case6| 3                             | 3                             | 1.78            | 43               | 0                | 101325          | 14               | 30               | 101325          |
| Case7| 12                            | 6                             | 6.1             | 43               | 0                | 101325          | 14               | 30               | 101325          |
| Case8| 6                             | 12                            | 5.42            | 43               | 0                | 101325          | 14               | 30               | 101325          |

Table 2: List of simulation cases.
Air can also provide sufficient heating and cooling capacity for cabin temperature control of the aircraft. Increasing the thickness of the insulation layer can reduce the heat loss (or heat gain) along the pipeline, but increasing the insulation thickness will also lead to the increase in the weight of the system. Therefore, it is necessary to study the effect of insulation thickness on the improvement of system insulation performance and the increase in system weight. In addition, the effect of the insulation layout on the thermal performance of the pipeline system is also investigated in this paper to find a method to optimize the thermal insulation performance of the system.

Table 2 lists the set of simulation cases considered here with a detailed thermal insulation thickness and the main boundary conditions. Cases 1-6 studied the effect of insulation thickness on the thermal performance of the air distribution system. Upstream and downstream pipelines in cases 7 and 8 employ different insulation layer thicknesses for a comparative study. The cabin ambient temperature \( T_{\text{amb}} \) is set at 0°C, the conditioned air temperature \( T_{\text{in}} \) of the mix manifold is 43°C, and the cabin pressure is set at 101325 Pa in cold day cases. The cabin ambient temperature \( T_{\text{amb}} \) is set at 30°C, the conditioned air temperature \( T_{\text{in}} \) from the mix manifold is 14°C, and the cabin pressure is set at 101325 Pa in hot day cases. The mass flow rate of each cooling pack is 1.22 kg/s, and the recirculation proportion is 22% in all cases cold days and hot days.

5.2.1. The Effect of Insulation Thickness. Figure 11 shows the thermal performance analysis results of the system in case 1 to case 6. It can be seen from the figure that the amelioration of the temperature drop and heat loss in the cold day case (the temperature rise and heat gain in the hot day case) with

| No. | Mass/kg | \( Q_h \)/kW | \( \Delta T_h \)/°C | \( Q_c \)/kW | \( \Delta T_c \)/°C |
|-----|---------|--------------|-----------------|--------------|-----------------|
| Case 7 | 6.1 | -12.56 | -6.43 | 4.7 | 3.4 |
| Case 8 | 5.42 | -13.44 | -5.52 | 5.04 | 2.82 |

Table 3: Comparison of the thermal insulation performance of case 7 and case 8.
the increase of thermal insulation thickness is not linear. This ameliorative effect is more significant as the thickness of the insulation layer is small, and this effect will gradually weaken and tend to be horizontal with a continuous increase in the thickness of the thermal insulation.

In the temperature pull-down case of hot days, the maximum temperature rise of all air outlets relative to the manifold temperature is $3.02\, ^\circ\!C$ when the thermal insulation thickness is $9\, \text{mm}$, which is close to the design requirements. In the temperature pull-up case of cold days, the maximum temperature drop of all air outlets relative to the manifold temperature is $−5.03\, ^\circ\!C$ when the thermal insulation thickness is $12\, \text{mm}$, which is close to the design requirements. The result also shows that the weight increase caused by the increase in the thickness of the insulation layer is not completely linear. With the increase in thickness, the system weight increase rate is rising up slightly, which means that the effect of the insulation thickness on the system weight is deteriorated. Therefore, it is unreasonable for the system design to achieve the highest thermal insulation performance by continuously increasing the thickness of the thermal insulation.

5.2.2. The Effect of Insulation Layout. If there are numerous thickness sizes of thermal insulation layers in the same system configuration, which are distributed in the upstream and downstream pipelines of the system, respectively. Case 7 arranges insulation of $12\, \text{mm}$ thickness on the upstream pipeline surface and insulation of $6\, \text{mm}$ thickness on the downstream pipeline surface, while case 8 covers the upstream pipeline with $6\, \text{mm}$ insulation and the downstream pipeline with $12\, \text{mm}$ insulation. As can be seen in Table 3, the total weight of the insulation in case 7 is $6.1\, \text{kg}$, which is slightly higher than the value of $5.42\, \text{kg}$ in case 8. The total heat exchange with the outside ambient of case 7 is $−12.56\, \text{kW}$ in the temperature pull-up condition and $4.7\, \text{kW}$ in the temperature pull-down condition, both of which have absolute values larger than those of case 8. However, the maximum temperature drop $ΔT_{th}$ of the conditioned air in case 7 is $6.43\, ^\circ\!C$ in the temperature pull-up cases, which is higher than the value of $5.52\, ^\circ\!C$ in case 8, and the maximum temperature rise $ΔT_{c}$ of the conditioned air in temperature pull-down cases is $3.4\, ^\circ\!C$ which is also higher than the value of $2.82\, ^\circ\!C$ in case 8.

Figure 12 shows the temperature contours of the air distribution pipeline system under the temperature pull-up condition of case 7 and case 8. As thicker insulations are assigned to the upstream pipeline for case 7 and to the downstream pipeline for case 8, the temperature drop of air flowing through the upstream pipeline of case 7 is smaller than that of case 8, making the temperatures of the upstream outlets of case 7 larger than those of case 8. However, it can
also be seen in this figure that the minimum air temperature of the downstream outlets of case 8 is higher than that of case 7 due to the thicker insulation of the downstream pipelines in case 8. This is because the air flow rate in the downstream pipeline is smaller than that in the upstream pipeline, leading to a larger temperature change under the same heat loss. Therefore, the variation in conditioned air temperature is more affected by the thermal insulation of the downstream pipeline of the air distribution system.

From the perspective of cabin temperature control, whether for temperature pull-up or pull-down cases, the design objective is to ensure that the maximum temperature drop (or temperature rise) from the mix manifold to each air supply outlet meets the requirements of the cabin temperature control system, which ensures that the conditioned air supply in each compartment area has sufficient heating (or cooling) capacity for regional temperature control. Case 8 has a lower value of the maximum temperature with lighter weight, and it is a better design compared with case 7.

6. Conclusions

The numerical method was investigated based on the combination of the 1D thermal fluid analysis program Flomaster and the 3D CFD program Fluent to achieve an efficient and accurate simulation of a passenger aircraft air distribution system with a large number of components. The effect of local irregular ducts on air distribution uniformity and the effect of pipeline insulation configuration on the temperature drop of the entire system are studied based on the simulation model. The conclusions are as follows:

1. The flow resistance of the local irregular ducts not only has an impact on the uniformity of the flow distribution of the compartment region, but also affects the total regional air mass flux. In this analysis, the existence of local irregular ducts increases the maximum relative deviation of flow distribution at the air supply point from 3.14% to 9.49%, exceeding the allowable 5% for the air distribution uniformity design.

2. The increase in thermal insulation thickness can decrease heat gain (or loss) through the pipe wall and the temperature change of conditioned air along the pipeline. However, this improvement gradually weakened, and the weight increase of the system accelerates. Considering the sensitivity of aircraft design to total weight, the thermal insulation layer should be as small as possible under the condition of satisfying the requirements of the overall temperature change of the conditioned air.

3. Adjusting the thermal insulation thickness layout can enhance the thermal insulation effect while keeping the total weight of the system from increasing. In this study, case 8 achieves 0.91 °C and 0.58 °C lower than case 7 for the absolute value of $\Delta T_h$ and $\Delta T_c$ under a lighter weight, respectively. Taking the temperature change of the conditioned air as the design object, it is more advantageous to arrange thicker insulation layers on the downstream pipelines.

Data Availability

The data used to support the findings of this study are included within the article.

Conflicts of Interest

The authors declare no conflict of interest.

Acknowledgments

This research was funded by Natural Science project of Hunan Province, China, grant number 2020JJ5393, and Education Department of Hunan Province, China, grant number 21B0618.

References

[1] Y. Tu and G. P. Lin, "Dynamic simulation of aircraft environmental control system based on Flomaster," Journal of Aircraft, vol. 48, no. 6, pp. 2031–2041, 2011.

[2] Z. Duan, H. Sun, C. Wu, and H. Hu, “Flow-network based dynamic modelling and simulation of the temperature control system for commercial aircraft with multiple temperature zones,” Energy, vol. 238, article 121874, 2022.

[3] Y. Tu and Y. Zeng, "One-dimensional and three-dimensional computational thermal fluid hybrid analysis-aided air distribution pipeline system design," Engineering Applications of Computational Fluid Mechanics, vol. 14, no. 1, pp. 436–446, 2020.

[4] L. Pang, J. Xu, L. Fang, M. Gong, H. Zhang, and Y. Zhang, "Evaluation of an improved air distribution system for aircraft cabin," Building and Environment, vol. 59, pp. 145–152, 2013.

[5] A. M. Farag and E. E. Khalil, "Numerical analysis and optimization of different ventilation systems for commercial aircraft cabins,” in IEEE Aerospace Conference Proceedings, pp. 1–12, Big Sky, MT, USA, 2015.

[6] Y. Zhang, J. Liu, J. Pei, J. Li, and C. Wang, "Performance evaluation of different air distribution systems in an aircraft cabin mockup," Aerospace Science and Technology, vol. 70, pp. 359–366, 2017.

[7] R. You, C. H. Lin, D. Wei, and Q. Chen, "Evaluating the commercial airliner cabin environment with different air distribution systems," Indoor Air, vol. 29, no. 5, pp. 840–853, 2019.

[8] Q. Cao, M. Liu, X. Li et al., "Influencing factors in the simulation of airflow and particle transportation in aircraft cabins by CFD," Building and Environment, vol. 207, article 108412, 2022.

[9] Y. Cui, "Study on the Simulink model for the testbed of the environmental control system of the aircraft,” Journal of Physics: Conference Series, vol. 1060, no. 1, article 012073, 2018.

[10] F. Meng, J. Sun, Y. Zhou, Z. Gao, and D. Cheng, “Modeling and validating of air cycle machine in aircraft environmental control system,” Huaxia Xuebao/CIESC Journal, vol. 71, pp. 328–334, 2020.
[11] I. Jennions, F. Ali, M. E. Miguez, and I. C. Escobar, "Simulation of an aircraft environmental control system," *Applied Thermal Engineering*, vol. 172, article 114925, 2020.

[12] Z. Shangguan and Q. Cheng, "Performance of a civil aircraft environmental control system coupled with hot water supply," *Applied Thermal Engineering*, vol. 203, article 117924, 2022.

[13] X. Chen, S. Yang, X. Sun, and L. Wang, "Research on 1D-2D co-simulation for cabin air environment accident," *Procedia Engineering*, vol. 121, pp. 1983–1989, 2015.

[14] Z. Xu, L. Yiwei, and X. Xu, "Performance simulation and improvement of new type aircraft environment control system base on AMESim software," *Aircraft Design*, vol. 38, no. 4, 2018.

[15] K. Chakinala, K. Sravanthi, and S. P. Jani, "CFD analysis of environmental control system for an aircraft," *Materials Today: Proceedings*, vol. 46, pp. 8502–8506, 2021.

[16] K. Marimuthu, K. Umanath, and S. Jai Krishnan, "Investigating the conjugate heat transfer phenomena on various ducts for aircraft environmental control system," *Materials Today: Proceedings*, vol. 46, pp. 3631–3638, 2021.

[17] Y. Zeng, B. Sun, and H. Sun, "Simulation calculation of flow characteristics of air separation system based on Flowmaster," *IOP Conference Series: Materials Science and Engineering*, vol. 470, no. 1, article 012023, 2019.

[18] R. L. M. Jiang, "Fault simulation for aircraft fuel system using Flowmaster," vol. 40, Springer, Berlin Heidelberg, 2014.

[19] Y. Tu, X. Xiao, and N. Li, "Computer analysis of large-scale aircraft landing gear retraction and extension control system," *Beijing Hangkong Hangtian Daxue Xuebao/Journal of Beijing University of Aeronautics and Astronautics*, vol. 39, no. 5, pp. 595–599, 2013.